Date April 30, 2000 Memo Number STI43:000430
Subject **ANSYS Tips & Tricks: Modeling Preloaded Fasteners**
Keywords Structural: Preload: PRETS179

1. **Introduction**:
There are many ways in which users can model preloaded fasteners in ANSYS. This memo describes three ways to accomplish this, including the use of the PRETS179 pretension element, which is available in ANSYS 5.6.[1]

2. **Background**:
The analysis of preloaded joints may be deemed necessary for design purposes and to satisfy specifications in design codes. There are several methods available to accomplish this, some of which are listed below:

- Constrain model at bolt locations to determine reaction forces. The tensile and shear loads could be checked independently with hand calculations.[2]
- The fastener could be split in half and constraint equations (CE) written to pull the two pieces together. The required restraint force could be determined. One can subsequently determine the required preload, which would dictate bolt selection.[3]
- Geometrically 1D (but spatially 3D) elements such as BEAM4 or LINK8 could be used with the initial strain option to connect two halves of a joint, pulling the halves together to produce the required preload.
- The fastener could be modeled with contact elements (node-to-node, node-to-surface, or surface-to-surface), and a geometric interference included, inducing a given preload.
- The fastener could be modeled without explicitly creating the geometric interference but adding the interference as a real constant. For example, when using the surface-to-surface contact elements (CONTA171-174), the real constant CNOF determines the initial interference. KEYOPT(9) is used to ramp the interference to produce a preload.
- The bolt could be given a "dummy" orthotropic coefficient of thermal expansion (ALPX). With the proper temperature load, the orthotropic thermal strain would "shrink" the bolt, producing a preload state.
- The joint could be modeled at the PRETS179 element used with the PTSMESH option to include a preload.

While this memo cannot cover all of the various techniques one can use in the analysis of fasteners, the last three methods of modeling preloaded joints will be reviewed. However, only the PRETS179 method will be covered in any detail. *Note that most of these methods require an iterative process to determine the appropriate input to produce the required initial strain (which induces the correct preload stress/force). The PRETS179 element, however, allows direct input of a preload force, which simplifies matters considerably.*

It is up to the engineer to determine the applicability of these methods to his/her situation. This memo assumes that the reader is familiar with the general analysis and selection of fasteners, including bolt size, grade, and proof strength. This memo will only serve to cover methods to model preloaded fasteners and assumes that the reader is familiar with appropriate engineering practices and the basics of the analysis of preloaded joints.

[1] Use of ANSYS 5.5 or 5.6 is assumed throughout this memo.
[2] This method usually does not include preload effects but is the simplest way to analyze fasteners. Hence it is included here for that reason.
[3] This method is similar to what the PRETS179 element does, although the latter performs it automatically.

3. **Use of Thermal Strain to Model Preload**:

One can use thermal strains to include a preload in a fastener. A "negative" temperature load will shrink the bolt, similar to the tightening effect. The basic procedure is as follows:

1) Assign *orthotropic* coefficients of thermal expansion to the fastener
2) Set appropriate values of reference temperature, either globally (TREF) or material-specific (MP,REFT)
3) Add contact/gap elements as needed
4) Apply a negative temperature to fastener
5) Solve first load step for preload effects only
6) Apply external loads and solve second load step

This method requires trial-and-error since one usually knows the preload force required for the fastener, not the required strain. Usually, a few tries will allow the user to get an approximate but satisfactory solution. Otherwise, an optimization loop can be used with the applied temperature as the design variable. The objective function would be the absolute difference between the required preload force and the current force in the fastener.

The attached input file "preload3.inp" demonstrates a simple axisymmetric example of a preload modeled by negative thermal strains ("shrinking" the bolt). Figure 1 shows a plot of SY after the applied temperature (preload). Figure 2 and 3 represent SY and SEQV plots of the system with externally applied loads.

4. **Use of Interference to Model Preload**:

Geometric interference, either explicitly modeled or with real constant values, can be used to induce preload effects in a fastener. The bolt is modeled to be "shorter" than required. During the interference fit solution, the bolt will consequently be "stretched" with an induced strain/load which is representative of the preload.

The node-to-node gap elements CONTAC12 and CONTAC52 use the INTF/GAP real constant whereas the surface-to-surface contact elements CONTA171-174 use the CNOF real constant. The use of the real constant interference value is recommended for reasons outlined below and will be the procedure discussed, along with CONTA171-174:

1) Add contact/gap elements as required
2) Assign interference/gap real constant for contact elements
3) Solve first load step with no loads applied
4) Apply external loads and solve

The surface-to-surface contact elements are recommended because of their powerful postprocessing features, compatibility with higher-order elements, and flexibility and ease-of-use. For CONTA171-174, KEYOPT(9) allows the user to ramp the initial interference (either geometric or CNOF).

This method, like the thermal strain technique, requires an iterative process to determine the appropriate value of CNOF which will result in the proper preload force. Hence, changing CNOF is much easier, especially for imported CAD geometry, than changing any explicitly modeled geometric interference. This technique lends itself well to design optimization, as described above, where the design variable is the CNOF real constant.

An example input file "preload2.inp" is provided to illustrate the use of this method on a simple model. Figure 4 presents a plot of SY after the interference fit solution (preload). Figures 5 and 6 illustrate the SY and SEQV results after the application of the external loads.

5. Use of PRETS179 to Model Preload:

In ANSYS 5.6, a new preload element has been introduced to make modeling of preloaded joints much easier. As noted in Sections 3 and 4 above, other techniques require a trial-and-error approach to determine an appropriate strain to induce the correct preload. With the PRETS179 element, this is not needed since a user directly inputs the preload as a force; this is information which is readily available to the analyst.

The PRETS179 element is described in detail in the Elements Manual. These elements are basically used to connect two "halves" of a bolt. A control node (also referred to as the "k" node) is used to apply loads and/or lock displacements. These elements can be created manually, but the recommended approach is to model the fastener as one normally would and use the PTSMESH command to automatically split the fastener elements and generate PRETS179 elements.

Usage/creation of the PRETS179 element is covered in Ch. 2.9 "Defining Pre-tension in a Joint Fastener" in the ANSYS 5.6 Basic Analysis Guide. One of the ways in which this element can be used is as follows:

1) Create model with contact elements
2) "Split" the bolt/fastener elements with the PTSMESH command
3) Apply the preload to the control "k" node of the PRETS179 elements
4) Solve preload load step
5) Delete force and constrain "k" node with current displacements (using %_FIX% value)
6) Solve load step[4]
7) Apply external loads and solve

In this manner, the preload (and subsequent change in bolt load) can be monitored with the reaction force at the "k" node. This makes /POST1 and /POST26 postprocessing quite easy.

It has been noted by some individuals that Steps 5 and 6 are not necessary but, in fact, wrong, when modeling a preloaded fastener. However, it is important to note why Steps 5 and 6 are required for most applications.

If the preload force at node "k" is left through the duration of the analysis, this means that ANSYS is applying a constant preload to the fastener. While, in some cases, this may be desired, usually the load in the fastener changes as external forces are applied.

On the other hand, removing the force at node "k" and applying an equivalent displacement with "D,node,ux,%_FIX%" locks the displacement and the preload. This is analogous to the physical system, where the tightening of a fastener reduces its length. Note that the displacement is applied *at the pretension section*, not across the entire bolt. This means that the rest of the fastener is free to stretch/shrink in response to the applied loads, and the load in the bolt will vary accordingly.

If the PRETS179 element is used in other situations, a constant pretension load may be deemed necessary, but, in the case of preloaded fasteners, the author believes that Steps 1-7 outlined above is the method analysts would want to use when employing the PRETS179 elements.

A simple input file, "preload1.inp", demonstrates the use of PRETS179 for the same problem outlined in Sections 3 and 4. Figures 7 shows SY after the initial preload. Figures 8 and 9 are plots of SY and SEQV after the externally applied loads. Figure 10 is a plot in /POST26 of the reaction force at the "k" node. (Question: Why does the bolt load dip down then increase? Hint: Rerun the "preload1.inp" input file after changing the plate radius R_TOPP to 1)

[4] Currently, the %_FIX% displacement value will be ramped from zero if included with the load step of externally applied loads. Hence, it is recommended to solve this constraint in a separate load step. Because of substitution of a force for the equivalent displacement, this load step should only take 1 iteration. Also, in the future, ANSYS will automatically recognize %_FIX% displacements and not ramp them as normal loads, so Step 6 will not be required.

6. Conclusions:

Three methods of modeling preloaded joints have been covered – namely, thermal strain method, interference fit with real constant CNOF, and the PRETS179 pretension element. If one examines the results from the attached input files which use all three methods, one will find that the answers obtained are very close, as shown in Table 1 below:

Table 1

	SY_A	SY_B	SY_C	SY_D	SY_{MIN}	SY_{MAX}
Thermal Strain Preload	42347	41342	-10587	-6565	-55326	56721
Interference Preload	42273	41271	-10566	-6552	-55103	56697
PRETS179 Preload	42276	41287	-10570	-6554	-55114	56661
Thermal Strain Final	43755	42625	-9513	469	-48337	57633
Interference Final	43693	42567	-9493	463	-48111	57608
PRETS179 Final	43685	42581	-9496	464	-48117	57558

Please note that the thermal strain method, while not too far off, would match the other two results more closely if the applied preload (imposed temperature BFE) was equal to the actual preload. The author did not use an optimization loop but performed two iterations to guess a reasonable applied temperature. Hence, there is some error introduced in the thermal strain method.

The main advantage of the PRETS179 element introduced at 5.6 is the fact that it does not require a trial-and-error process to model the preload, so this is the recommended method in modeling preloaded fasteners (see Section 5).

As with any capability of ANSYS, if the user is planning to implement any feature which is unfamiliar/new to him/her, it is strongly recommended to test out the modeling methods and practice postprocessing of results on a smaller, more manageable model before utilizing it for production work. CSI also offers one-on-one mentoring sessions to help a user become proficient with any feature of ANSYS and to help improve one's modeling practices in general.

Sheldon Imaoka
Collaborative Solutions, Inc. (LA Office)
Engineering Consultant

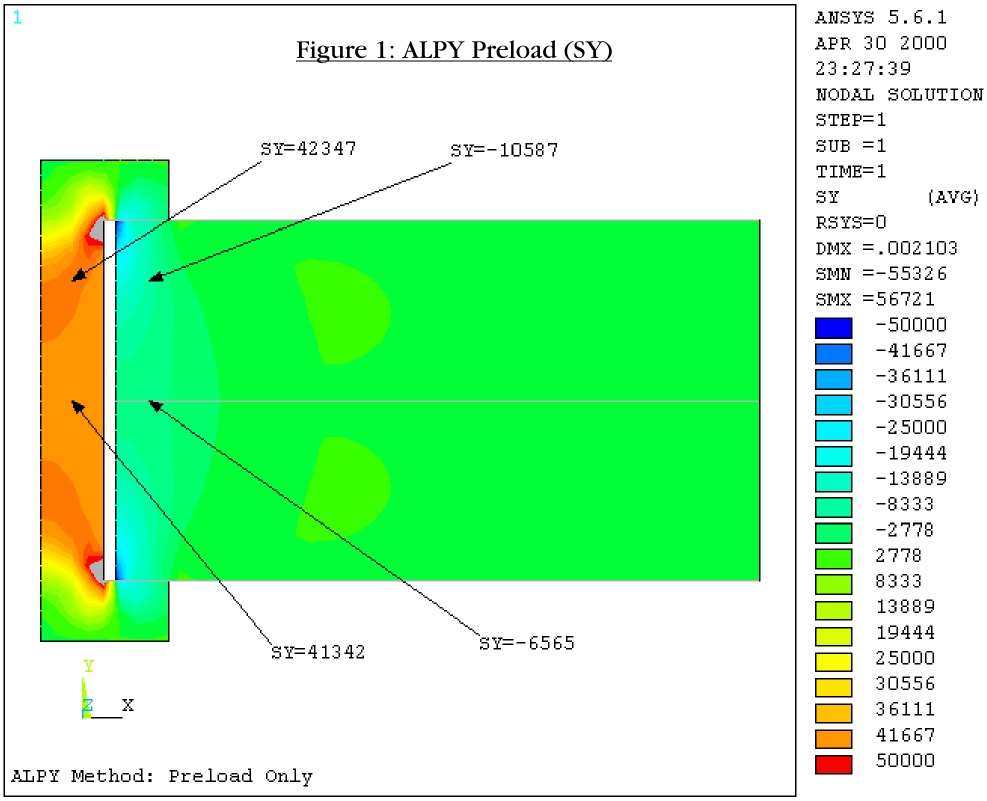


Figure 1: ALPY Preload (SY)

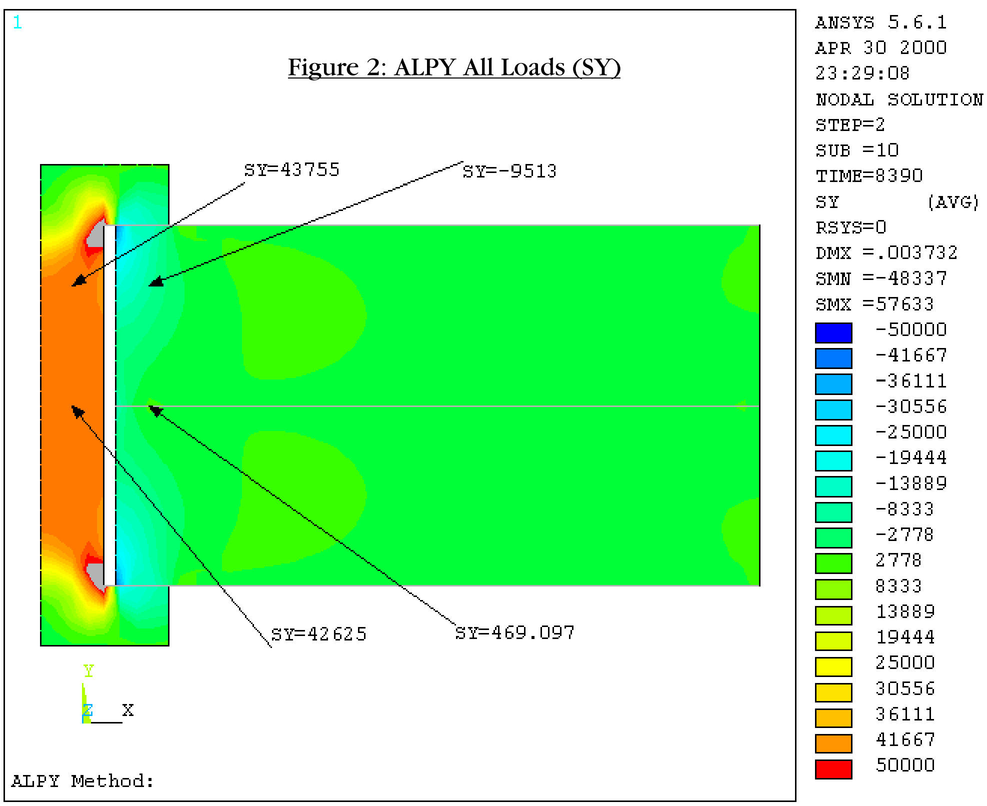


Figure 2: ALPY All Loads (SY)

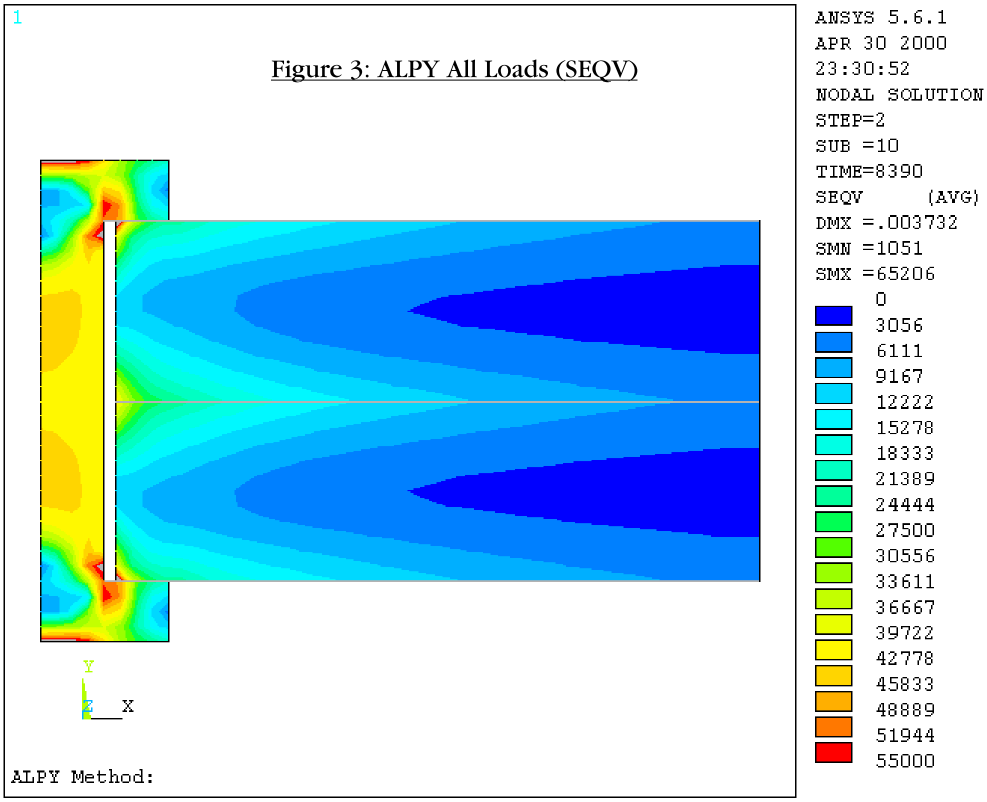
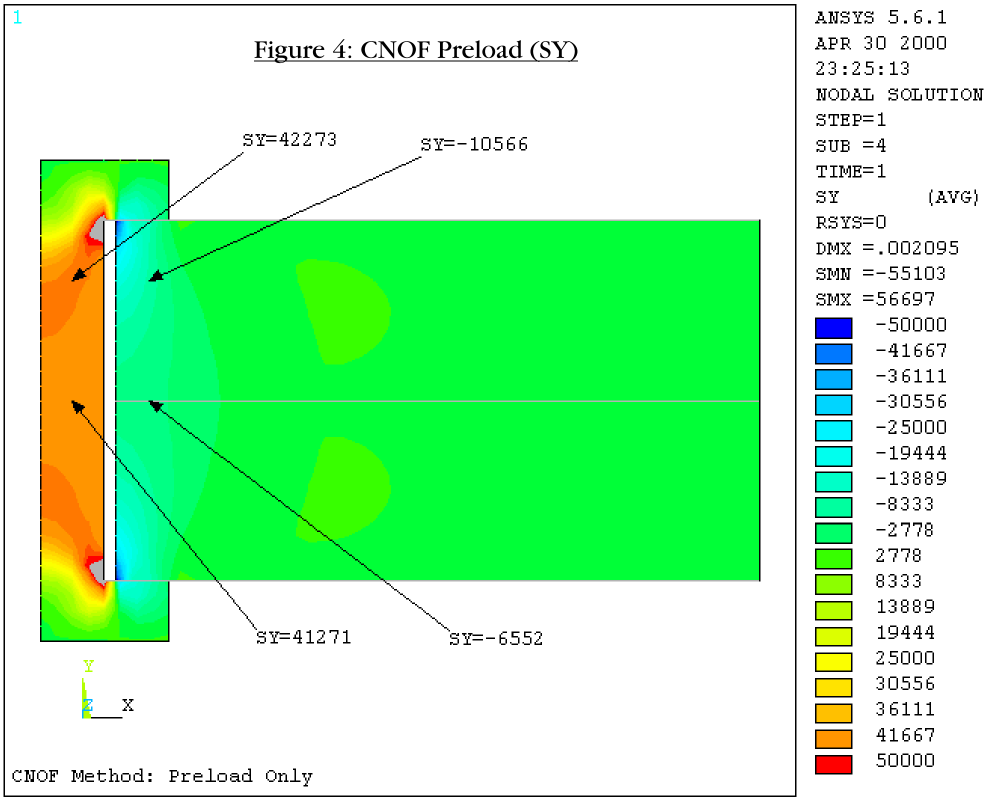
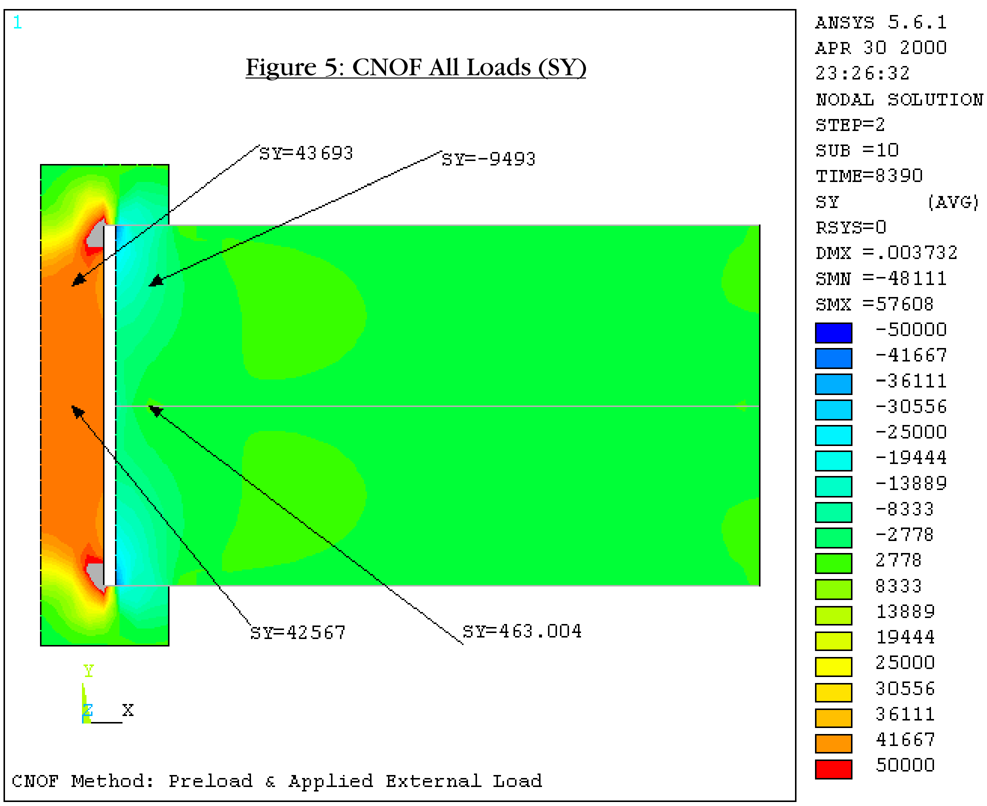


Figure 3: ALPY All Loads (SEQV)


1
Figure 4: CNOF Preload (SY)
SY=42273
SY=-10566
SY=41271
SY=-6552
Y
Z
X
CNOF Method: Preload Only
ANSYS 5.6.1
APR 30 2000
23:25:13
NODAL SOLUTION
STEP=1
SUB =4
TIME=1
SY (AVG)
RSYS=0
DMX =.002095
SMN =-55103
SMX =56697
-50000
-41667
-36111
-30556
-25000
-19444
-13889
-8333
-2778
2778
8333
13889
19444
25000
30556
36111
41667
50000


1
Figure 5: CNOF All Loads (SY)
SY=43693
SY=-9493
SY=42567
SY=463.004
Y
Z
X
CNOF Method: Preload & Applied External Load
ANSYS 5.6.1
APR 30 2000
23:26:32
NODAL SOLUTION
STEP=2
SUB =10
TIME=8390
SY (AVG)
RSYS=0
DMX =.003732
SMN =-48111
SMX =57608
-50000
-41667
-36111
-30556
-25000
-19444
-13889
-8333
-2778
2778
8333
13889
19444
25000
30556
36111
41667
50000

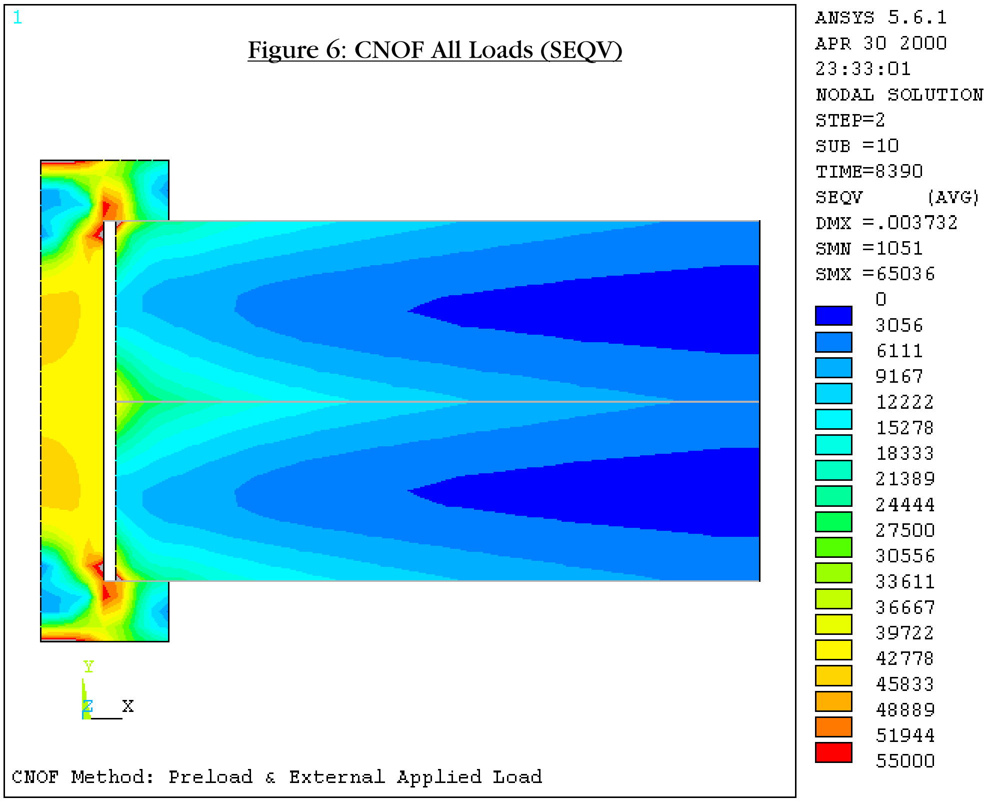


Figure 6: CNOF All Loads (SEQV)

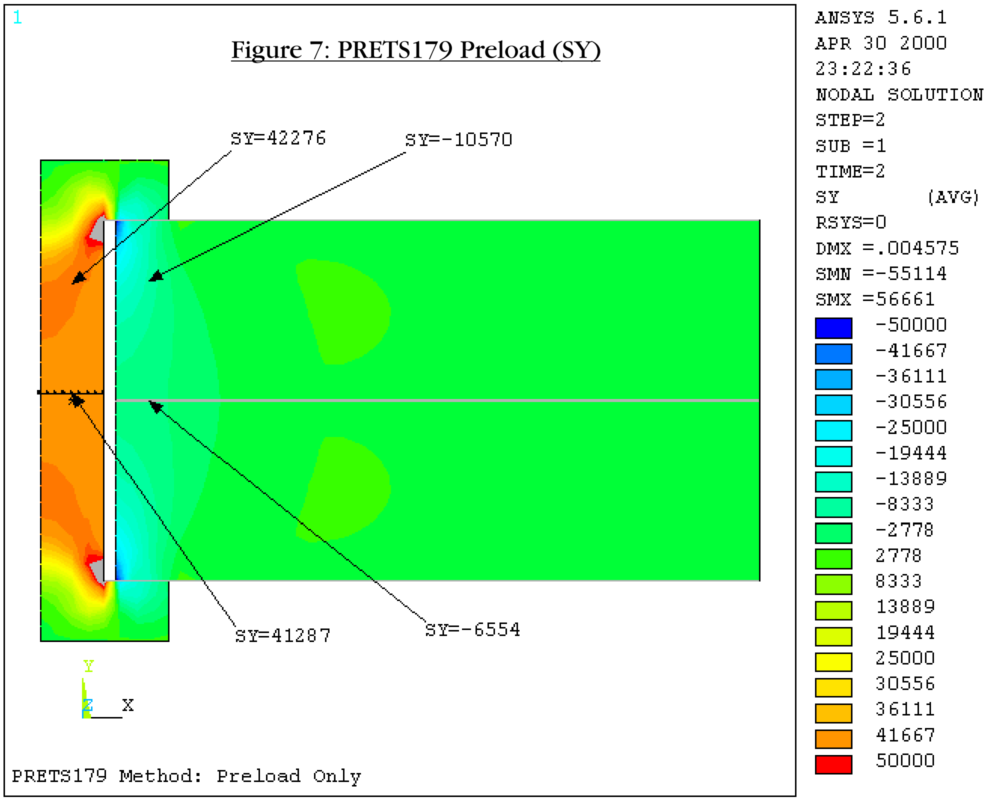


Figure 7: PRETS179 Preload (SY)

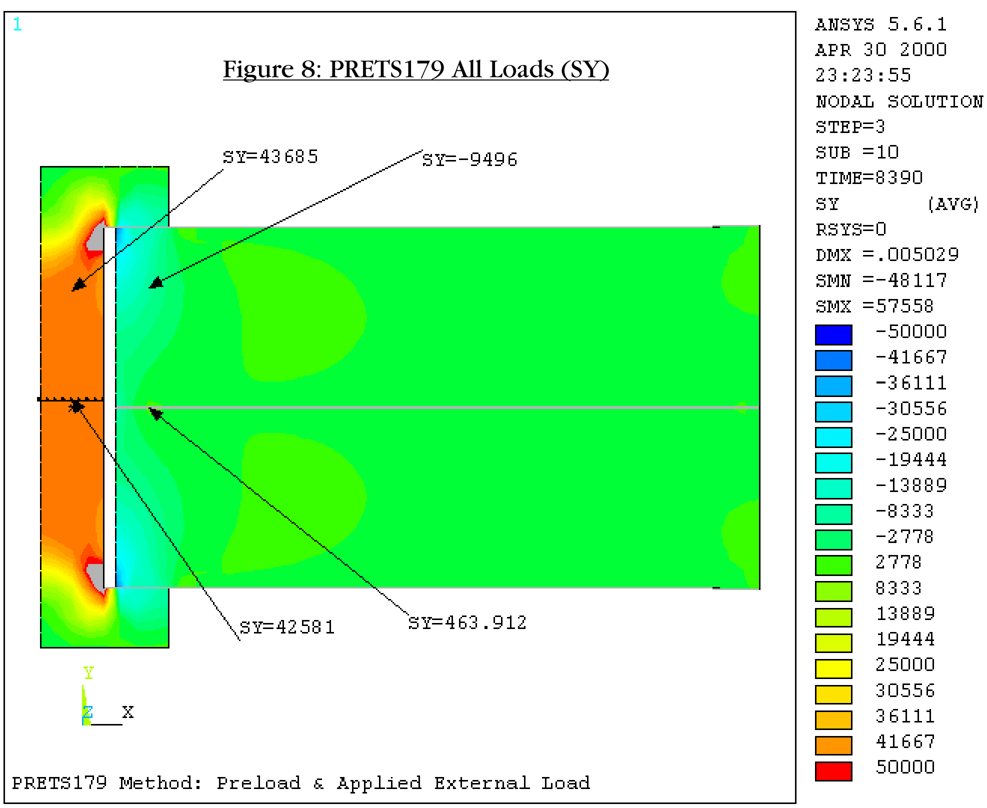


Figure 8: PRETS179 All Loads (SY)

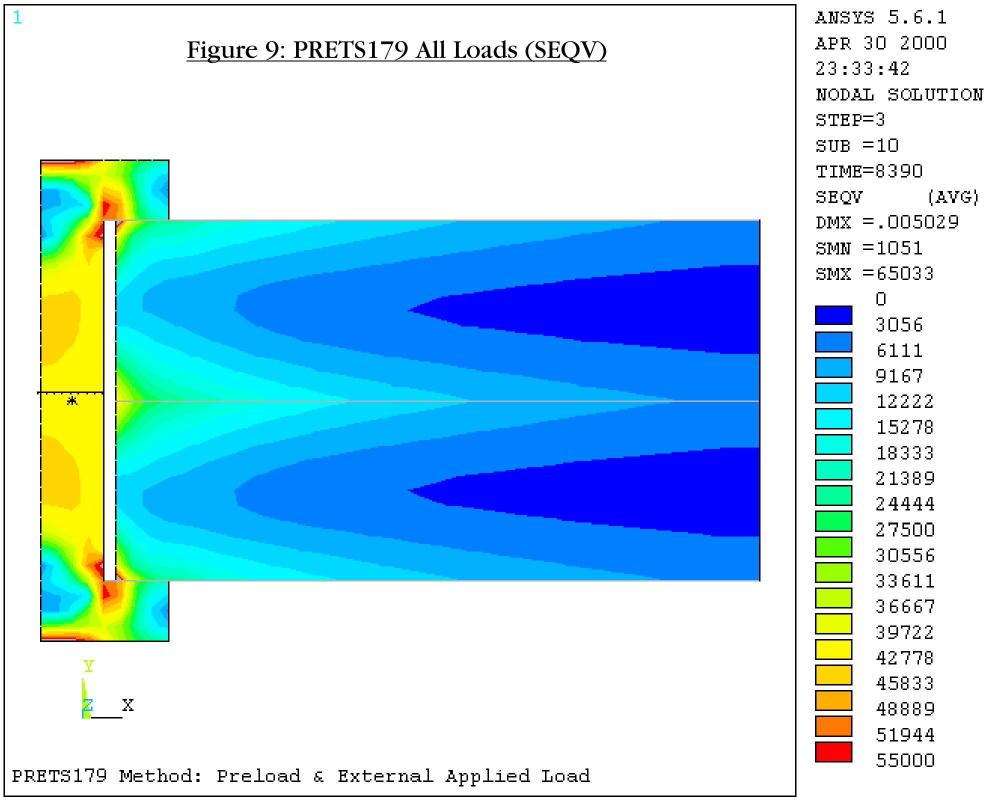


Figure 9: PRETS179 All Loads (SEQV)

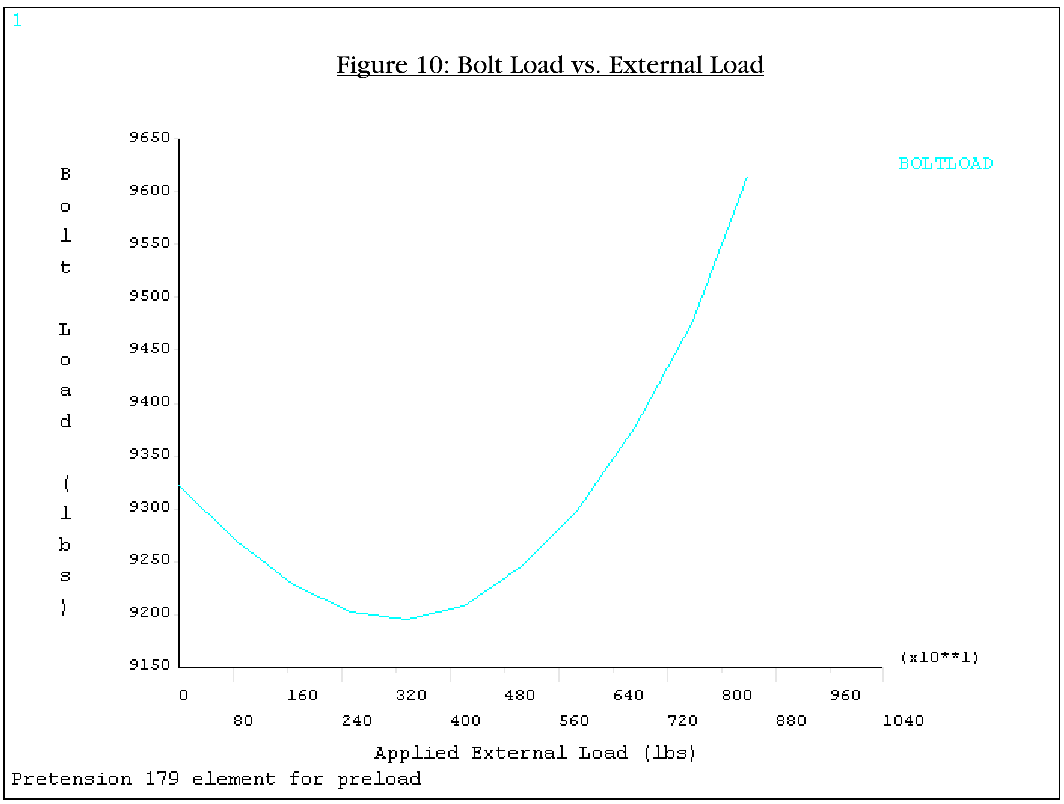


Figure 10: Bolt Load vs. External Load

ANSYS Tip of the Week

"ANSYS Tip of the Week" (TOTW) is provided for customers of Collaborative Solutions, Inc. (CSI) with active TECS agreements, distributed weekly in Adobe Acrobat PDF format via email. Unless otherwise stated, information contained herein should be applicable to ANSYS 5.4 and above, although usage of the latest version (5.6 as of this writing) is assumed. Users who wish to subscribe/unsubscribe or to view older TOTW archives can visit

`http://www.csi-ansys.com/tip_of_the_week.htm`

Corrections, comments, and suggestions are welcome and can be sent to operator@csi-ansys.com [they will be distributed to the appropriate person(s)]. While CSI engineers base their TOTW on technical support calls and user questions, ideas on future topics are appreciated. Users who wish to submit their own TOTW are encouraged to do so by emailing the above address for more information.

XANSYS Mailing List

The XANSYS mailing list is a forum for questions and discussions of the use of ANSYS. As of 04/00, there are more than 1000 subscribers with topics ranging from Structural, Thermal, Flotran, to Emag analyses, to name a few. Users are encouraged to subscribe to evaluate the usefulness of the mailing list for themselves. Also, either (a) using the mail program to filter [xansys] messages or (b) using the "digest" option to receive one combined email a day is strongly recommended to minimize sorting through the volume of postings.

```
This list is for *ALL* users of the ANSYS finite element analysis program from around the
world. The list allows rapid communication among users concerning program
bugs/ideas/modeling techniques. This list is NOT affiliated with ANSYS, Inc. even though
several members of the ANSYS, Inc. staff are subscribers and regular contributors.

To SUBSCRIBE: send blank email to xansys-subscribe@onelist.com
To unsubscribe send blank email to xansys-unsubscribe@onelist.com
Archived on http://www.infotech.tu-chemnitz.de/~messtech/ansys/ansys.html
ANOTHER archive on http://www.eScribe.com/software/xansys/
(A poor archive is also at http://www.onelist.com/archives.cgi/xansys)
```

CSI ANSYS Technical Support, Training, & Mentoring

Collaborative Solutions, Inc. is committed to providing the best customer support in our industry. Three people will be devoted to technical support from 8:00 a.m. to 5:00 p.m. PST every working day. CSI customers with active TECS (maintenance) agreements may contact CSI by any of the following ways:

```
Phone:  760-431-4815                             WWW: http://www.csi-ansys.com
Fax:    760-431-4824                             FTP:  ftp://ftp.csi-ansys.com
E-mail: firstname.lastname@csi-ansys.com
```

CSI Engineers:

Karen Dhuyvetter	Greg Miller	Sean Harvey	Alfred Saad
Bill Bulat	Sheldon Imaoka	David Haberman	Mike Rife

CSI believes strongly in the value of training and mentoring to help make customers successful using ANSYS. Training classes are usually 2-3 days in duration and provide instruction on various topics, including structural nonlinearities, heat transfer, and dynamics. Mentoring sessions involve working with a CSI engineer one-on-one on specific projects. These sessions help reinforce applicable subject matter covered in training classes or help ensure that the customer is using ANSYS most efficiently and effectively.

Training class schedules are posted at: `http://www.csi-ansys.com/training.htm`

Please contact your account manager for more details on training, mentoring, consulting, and other CSI services.