Boeing Computer Services
P.O. Box 24346
Seattle, WA 98124-0346

June 14, 1994
G-1151-RSO-94-176

Document Control Desk
United States Nuclear Regulatory Commission
Washington, D.C. 20555

BOEING

Reference: a) Boeing Letter G-1551-RSO-365 dated August 31, 1992; R. S. Orr to the NRC Operations Center

b) NRC Letter Docket No. 99901227 dated August 12, 1992; L. J. Norrholm to R. S. Orr; Subject: Response to 10 CFR 21 Inquiry

Dear Sir or Madam:

In accordance with the reference correspondence and 10 CFR 21, Boeing is sending the NRC the attached error notice(s) received from our former software suppliers. Because of unknown current addresses, the following former customers were not notified:

Reactor Controls, Inc.

Echo Energy Consultants, Inc.

Nuclear Applications and Systems Analysis Company (Japan)

Nuclear Power Services

Error notices have been sent to our other former customers.

Very truly yours,

J Francalanza/for

R. S. Orr
Nuclear Administrator
G-1151 M/S 7A-33
(206) 865-6248

Attachment(s): Class3 Error Reports 94-21 and 94-22. Also ANSYS QA Notice QA94-07

IE09
1/1

ANSYS® CLASS3 ERROR REPORT

ERROR NO: 94-21

KEYWORDS: *ELEM93* *SHELL93* *CONJUGATE GRADIENT* *SUBSPACE*

DESCRIPTION OF ERROR:

Curved shell models constructed with SHELL93 elements will behave too stiffly when:

1. The .FULL file is used (either conjugate gradient method (EQSLV,JCG) for static analysis, or subspace for modal analysis (ANTYPE,MODAL with MODOPT,SUBSP).

and 2. The model has a radius of curvature (ρ) to element thickness (t) ratio (ρ/t) greater than approximately 2500:1.

FIRST INCORRECT VERSION(S):*	CORRECTED IN:*
Rev. 5.0	Rev. 5.1

SUGGESTED USER ACTION FOR RUNNING ON UNCORRECTED VERSION:

Use the frontal solver (EQSLV,FRONT) [default] for static analysis and the reduced method (MODOPT,REDUC) [default] for modal analysis on models that meet the above criteria.

COMMENTS:

In Rev. 5.1, the ρ/t ratio where overly-stiff models can occur has been increased to approximately 20000:1 for the JCG and full subspace solvers. Other solvers have no known limits.

Note: Plate models that are exactly flat are not affected by this error.

AUTHOR/CORRECTOR: _____ DATE: June 1, 1994
Anton V. Mobley

REVIEWED BY QA: _____ DATE: June 1, 1994
Mark C. Imgrund

APPROVAL: _____ DATE: June 1, 1994
John A. Swanson

* If a product name is not included in the "first incorrect version", the full ANSYS program is implied. For products not listed, this error does not apply, but see the reverse side for equivalent product designations.

Unless noted otherwise, this error report also applies to all revisions after the first incorrect one and prior to the corrected revision. All revisions after "corrected in" are corrected. Manual corrections are included in on-line documentation as appropriate. Please see the reverse side of this sheet for additional information on ANSYS revision identifiers.

FORM SASI-QA25
DEC 10 1990

ANSYS® CLASS3 ERROR REPORT

ERROR NO: 94-22

KEYWORDS: *ELEM10* *LINK10* *LUMPM*

DESCRIPTION OF ERROR:

LINK10 (the tension-only or compression-only spar element) has the following errors:

1. When using the lumped mass matrix option (LUMPM,ON), only 2/3 of the mass is included. This is because the mass matrix erroneously remains a consistent matrix, with the off-diagonal terms ignored.

2. When the status of the element is slack (STAT=2) and there should be no stiffness assigned to a slack element (KEYOPT(2)=0) [default], the stiffness is erroneously not set to zero.

3. For the first cumulative iteration, the prestrain (real constant ISTRN) is improperly used for cases when the prestrain represents a slack element. This will cause incorrect results for cases where only one iteration is allowed, such as a static preload situation before a dynamic analysis.

FIRST INCORRECT VERSION(S):* CORRECTED IN:*

Rev. 5.0 Rev. 5.1

SUGGESTED USER ACTION FOR RUNNING ON UNCORRECTED VERSION:

For each of the three LINK10 errors, take the following steps:

1. If using LUMPM,ON, then increase density (MP,DENS) by a factor of 1.5.

2. Use either KEYOPT(2)=1 or KEYOPT(2)=2.

3. Force the execution of at least two iterations before results are to be used. This may be done using an extra SOLVE command during the static preload.

COMMENTS:

AUTHOR/CORRECTOR: Peter Kohnke DATE: June 1, 1994

REVIEWED BY QA: Mark C. Imgrund DATE: June 1, 1994

APPROVAL: John A. Swanson DATE: June 1, 1994

*If a product name is not included in the "first incorrect version", the full ANSYS program is implied. For products not listed, this error does not apply, but see the reverse side for equivalent product designations.

Unless noted otherwise, this error report also applies to all revisions after the first incorrect one and prior to the corrected revision. All revisions after "corrected in" are corrected. Manual corrections are included in on-line documentation as appropriate. Please see the reverse side of this sheet for additional information on ANSYS revision identifiers.

FORM SASI-QA25
DEC 19, 1988

ANSYS QA NOTICE

NOTICE NO: QA94-07

SUBJECT: *CREEP*

DESCRIPTION:

Built-in creep equations for nuclear steels (C_6 = 9, 10 and 11) have not been updated since the 1977 revision of the Nuclear Systems Materials Handbook. We understand that these equations are obsolete, but we have not found a reliable source for the new equations. Any users with more current information are requested to notify us (Cecil Rogers, crogers@swanson.com) for a possible upgrade of our equations.

Meanwhile, users are advised to use the USER CREEP facility (refer to Appendix B.1 of the Procedures Manual) to supply the desired creep equations (if known).

AFFECTED VERSIONS: All since Rev. 4.0

AUTHOR/CORRECTOR: Cecil R. Rogers DATE: June 1, 1994

REVIEWED BY QA: Mark C. Ingrund DATE: June 1, 1994

APPROVAL: John A. Swanson DATE: June 1, 1994

SASI-QA3
AUG. 29, 1993