NMC
Committed to Nuclear Excellence

Point Beach Nuclear Plant
Operated by Nuclear Management Company, LLC

June 3, 2005

NRC 2005-0068
10 CFR 50.90

U.S. Nuclear Regulatory Commission
ATTN: Document Control Desk
Washington, DC 20555

Point Beach Nuclear Plant Unit 2
Docket 50-301
License No. DPR 27

Submittal of Supporting Analyses Regarding Control of Heavy Loads

References: 1. NMC Letter to NRC Dated May 13, 2005

In Reference 1, Nuclear Management Company, LLC (NMC), requested exigent review and approval, in accordance with the provisions of 10 CFR 50.90 and 50.91(a)(6), of a proposed amendment to the license for Point Beach Nuclear Plant (PBNP), Unit 2, to support a change to the PBNP licensing basis regarding control of heavy loads on a one-time basis for the upcoming lift of the Unit 2 reactor vessel head (RVH).

As discussed during a telephone conference between NMC personnel and Nuclear Regulatory Commission (NRC) staff on June 3, 2005, NMC is providing initial results of finite element analyses (FEAs) of the postulated RVH drop scenario prepared by Sargent & Lundy and also by Automated Engineering Services Corporation.

These two independently performed analyses, which have been reviewed by NMC personnel, are consistent and demonstrate that reactor vessel head deflection following a postulated drop would be less than four inches. This amount of deflection would be insufficient to cause a loss of decay heat removal capability.

Enclosure 1 provides Sargent & Lundy Report, "Analysis of Postulated Reactor Head Load Drop onto the Reactor Vessel Flange Initial Summary Report", received June 3, 2005. Enclosure 2 provides Automated Engineering Services Corporation Report, "Summary Report on the 3-D Finite Element Analysis of the Reactor Vessel Head Load Drop for PBNP", dated June 3, 2005. The final results of the FEAs are planned to be submitted in support of Reference 1 the week of June 6, 2005.

6590 Nuclear Road • Two Rivers, Wisconsin 54241
Telephone: 920.755.2321

A001

This letter contains no new commitments or changes to existing commitments.

Dennis L. Koehl

Dennis L. Koehl
Site Vice-President, Point Beach Nuclear Plant
Nuclear Management Company, LLC

Enclosures (2)

cc: Regional Administrator, Region III, USNRC
Project Manager, Point Beach Nuclear Plant, USNRC
Resident Inspector, Point Beach Nuclear Plant, USNRC

ENCLOSURE 1

SARGENT & LUNDY REPORT,
"ANALYSIS OF POSTULATED REACTOR HEAD LOAD DROP ONTO THE REACTOR VESSEL FLANGE INITIAL SUMMARY REPORT",
RECEIVED JUNE 3, 2005

(13 pages follow)

Sargent & Lundy LLC

Analysis of Postulated Reactor Head Load Drop onto the Reactor Vessel Flange

Initial Summary Report

Prepared for the Nuclear Management Company's (NMC)
Point Beach Unit 2 Nuclear Generating Plant

1. Purpose and Scope

The Nuclear Management Company (NMC) has requested that Sargent Lundy LLC (S&L) perform an analysis to consider the effect of 26'-5"vertical drop of reactor vessel (RV) head onto RV flange in Unit 2 of Point Beach Nuclear Plant. The analysis is performed using the ANSYS computer program. The elements in the impact path have been modeled as necessary, and their response has been evaluated. The methodology used for the analysis is similar to what was used for the Prairie Island head drop analysis. Also, NRC questions asked on the Prairie Island analysis were considered in performing this analysis.

This report is considered as an initial summary report because it is based on work in progress, which has been reviewed for overall adequacy of methods, inputs, and reasonableness of results. This report is not a summary of an approved calculation. The final approved calculation will be consistent with the engineering methods discussed herein, but the actual numerical values for some inputs and results may vary from those presented here. These likely changes are not anticipated to affect the overall conclusions that are presented herein.

2. Description of Load Path

Refer to drawings 233-681, 233-682, and C-2320. When reactor vessel (RV) flange is vertically impacted by the falling head, the impact effect is directly transmitted through the vessel to four RV nozzle supports and two RV bracket supports to a framework of six horizontal box girders.

Each girder supports one reactor support at its center, and it is supported at its ends by 12" Schedule 120 pipe columns. Refer to drawing C-2320. The top of girder elevation is 31'-9 3/8". The bottom of girder is at elevation 30'-1 7/8". The columns are pinned at bottom, and anchor bolts anchor the base plate to concrete base mat at elevation 10'-0".

Refer to drawings C-325, C-326, and C-135. Each pipe column passes through a hole in a concrete shelf structure. The elevation of top of the shelf is 29'-10 1/2". The bottom of the shelf elevation is 14'-1 ¾". A vertical construction joint exists between the outer surface of the 3'-2" thick shelf structure and the inner face of biological shield structure. The shelf structure has vertical reinforcement at its inner face, and two rows of horizontal bars near the top and bottom ends of the shelf that provide shear-friction resistance against vertical movement of the shelf relative to the biological shield structure. The inner face of the shelf structure has a ¼" liner plate.

Noting that the bottom of girder elevation is 30'-1 7/8", and top of shelf elevation is 29'-10 1/2", it is seen that if impact causes the RV move downward in excess of 3 3/8", the girders will vertically impact the top of the shelf structure. (At the location of girder ends the gap is much smaller than the girder center because of column cap plate connection. However, the bearing area is small. Concrete will locally crush, allowing full girder contact with the shelf structure after 3 3/8"of girder downward movement.)

As seen on drawing C-326, the gap between the hole for column in shelf structure and the column is about 1 ¾". Therefore, in the event of initial column buckling, the shelf structure can provide lateral support for the columns. The lateral deflection of the column is calculated to determine the adequacy of lateral support from the shelf structure.

It should be noted that at elevation 17'-1 ¾", a ½" horizontal plate connects the columns to the shelf structure with zero gap (Detail 1 in drawing C-326).

In summary, the load path consists of the RV, RV supports at four nozzles and two brackets, six support girders, and six pipe columns and their pins and base plates and concrete foundation. If the vertical displacement of girders exceeds 3 3/8", the shelf structure also gets impacted transferring the vertical load through shear-friction to biological shield wall.

3. Method of Analysis

This section describes the analysis models, solution process and acceptance criteria.

3.1 Analysis Models

Analysis models consist of static analysis models for stiffness calculations and a dynamic impact model. The static analysis models are used to construct a static load-displacement diagram for the RV Inlet/Outlet nozzles, RV support brackets and girder frame with steel column supports. Section 3.1.1 describes the element types and properties of this model.

The dynamic transient model consists of a two-mass model with springs, dash-pot and gap in a vertical configuration. The top mass represents the falling head, and the bottom mass represents the target RV model supported by various springs and gap which represent the stiffness of the nozzle/bracket support, the girder box frame/column supports and the concrete shelf structure. Section 3.1.2 describes the parameters of this impact analysis model.

3.1.1 Static Analysis Models

ANSYS SHELL181, four node finite strain elements are used in FEA models for static load-deflection analyses. Non-linear material properties are modeled with a strength increase factor of 10% to account for the strain rate effects due to the dynamic impact. Large deformation analysis option was selected to account for potential buckling and yielding in the structural components along the impact load path.

3.1.1.1 RV Nozzle Stiffness:

Figure 3-1 shows the FEA model for the RV Inlet and Outlet nozzles. A true stress-strain curve from Battelle Pipe Fracture Encyclopedia, U.S. Nuclear Regulatory Commission, 1997, which has similar yield strength (Sy) and ultimate strength (Su) of the nozzle material (ASTM A-508-64 Class 2), was used in the analysis. The load-deflection curve of the nozzle is shown in Figure 3-2.

3.1.1.2 RV Bracket Support Stiffness:

Figure 3-3 shows the FEA model for the RV bracket support. The material properties of the bracket were assumed to be the same as that of the nozzles. The load-deflection curve of the nozzle is shown in Figure 3-4.

3.1.1.3 Girder Box Frame and Support Column Stiffness:

Figure 3-5 shows a 60-degree sector FEA model for the girder box frame and a column support. Load is applied at the mid-span of the girder box beam (the symmetric boundaries of the model). The support column base is a pin connection and the column top plate is bolted to the girder box beam connection. The column may buckle under compressive load and come in contact with the surrounding concrete shelf. Between the elevations 14'-1 3/4" to 17'-1 3/4", the gap between column and surrounding concrete is 1.75". However, there are channels and openings in the concrete shelf inner surface at elevations above 17'-1 3/4", therefore, only the outer half of surrounding concrete is modeled. Figure 3-5 shows the lateral gap elements along the support column. Note that the 1/2" plate at elevation 17'-1 3/4" provides a lateral support for the column without gap. The girder box material is A-517 Type F, a high strength steel with Sy =90 ksi and Su =110 Ksi. A similar true-stress true-strain curve from the material database was modified to match with the Sy and Su of A-517 Type F material. The true-stress true-strain of the support column material (A-53 Grade B) is available from the material database. The load-deflection curve of the girder box frame support column is shown in Figure 3-6.

3.1.1.4 Concrete Shelf Vertical Stiffness and Load Capacity:

The 3'-2" thick shelf structure is 3 3/8" below the bottom of the girder box beam and is potentially impacted by the girder beam. The initial stiffness, load capacity, and deformation limit of the shelf structure are shown in Figure 3-7.

Vertical load carrying capacity of the concrete shelf is determined considering the ultimate shear capacity along the vertical cylindrical construction joint plane between the concrete shelf and the biological shield wall. This ultimate shear capacity is determined per Section 11.7 of the ACI 349 using a friction coefficient of 1.0. The selection of friction coefficient of 1.0 is considered justified due to presence of shear keys at the top and bottom of the construction joint plane.

The load deformation characteristic of the concrete shelf is determined considering half space spring constant modified by the vertical deflection of the concrete shelf at initiation of yielding of the shear friction rebars. In addition, the Dynamic Strength Increase Factors (DIF) per Section C.2.1 of the ACI 349 are used to account for the impactive nature of the load. The concrete strength at the present time is determined considering the increase due to aging of the concrete as well as the actual concrete pour strength of 5200 psi. It should be noted that no credit is taken for the additional energy absorption capacity of the liner.

Using the guidelines of Section C.3.3 of the ACI 349, the limiting deflection for the concrete shelf is determined considering a ductility of 10 because the load carrying capacity of the concrete shelf is controlled by the yielding of the shear friction rebars. Based on the current analysis results, the required ductility demand is found to be less than 4.

3.1.2 Impact Analysis Model

The impact analysis model is a vertical model consisting of a single mass representing the falling head, and a second single mass that represents the modeled RV target assembly. Figure 3-8 depicts this model; M1 represents the 194,000 lb falling mass of the vessel head and M2 represents the 707,600-lb reactor vessel target including fuel, internals and water.

Contact stiffness K1 at the RV flange and the head on impact is taken as 100 times higher than the stiffness of the target RV and its supports such that it is nearly rigid contact. An impact-damping factor of 5% of the critical damping is considered in the analysis, which is the same damping factor used in the Prairie Island Nuclear Station head drop analysis. It should be noted that because of a softer supporting structure at Point Beach, this damping value is considered conservatively low. The spring K1 and the dashpot C1 (5% damping) are modeled using ANSYS COMBIN40 elements with an initial gap equal to 26.4 feet.

There are two parallel non-linear spring elements connected to M2. Spring K4N represents four RV nozzles, and spring K2B represents two RV bracket supports. These springs are connected to spring K6G, which represents the girder box frame and support columns. These non-linear springs are modeled with the ANSYS COMBIN39 based on the nonlinear load-displacement diagrams obtained from analysis model described in Sections 3.1.1, 3.1.2 and 3.1.3.

The concrete shelf vertical stiffness is modeled using ANSYS COMBIN40 elements with an initial gap equal to 3 3/8", a support stiffness of 6* 28.925 E+6 lbf/in, and a sliding force of 6*1,414,000 pounds.

3.2 Solution Process

The static analysis models are executed to define the non-linear load displacement curves which define the effective stiffness of the supporting structures and are used in the impact analysis model.

The impact analysis model is executed with the dynamic transient starting at time $t = 0$ for a specified length of time with small time steps. If the results indicate a rebound of the head mass, then the analysis time duration is lengthened and the analysis is again executed. This process is iterated until the rebounding of the head mass essentially stops. The maximum displacements and reaction forces reported are checked to see that the small amplitude rebounds are not resulting in increased loading or displacement. At the point when rebounding no longer increases either loading or displacement, a solution is arrived at.

3.3 Acceptance Criteria

In addition to the requirement that the supporting structure should be capable of resisting the impact load, the following specific acceptance criteria shall be met:

1. After its contact with the flange, the dropped head should undergo an upward velocity. Subsequent strikes and rebounds of the head must be associated with noticeably decreased rebound amplitudes approaching nearly zero rebound.

2. The maximum vertical deflection of the concrete shelf must not exceed the deflection limit obtained from a ductility of 10 on the steel re-bars.

3. In order for the concrete shelf to provide lateral support for the supporting columns, the lateral force must not exceed the concrete shelf lateral load capacity.

4. Results

The analyses described above have been performed on a preliminary basis, and have been re-performed to demonstrate repeatability. An overall review of methodologies, inputs, and the reasonableness of conclusions drawn from the analysis results have been performed. A detailed, complete numerical check of the work has not been performed.

The analysis results predict a maximum vertical displacement of the reactor vessel after impact of 3.55 inches. Figures 4.1 and 4.2 depict the results of this analysis showing the RV Head and vessel flange time-displacement curves, respectively. It should be noted that the approximately 40" rebound shown in Figure 4.1 will be significantly reduced if a more realistic damping value is used in the analysis. Figure 4.3 shows the stress and the deformed shape of the column supports. Stresses in the support bracket and nozzle are shown in Figures 4.4 and 4.5 respectively.

The total impact load on the support columns is about 11,020,000 pounds or 1,836,667 pounds per column. The total vertical load on the concrete shelf structure is 8,484,000 pounds or 1,414,000 pounds per impact location, which is the same as the plateau of concrete shelf load-deflection limit. The total deflection of the concrete shelf is 0.18", which is less than the deflection limit of 0.517".

The concrete shelf is also required to provide lateral support for the stability of the support columns located within the shelf. Between elevations 17'– 1 ¾" and 29'– 10 ½", the concrete shelf will be subjected to lateral stability loads directed away from the reactor centerline towards the biological shield wall. Thus, the concrete shelf will simply transfer these loads to the biological shield wall in bearing. By engineering judgment, the biological shield wall is considered adequate for these minor loads without any detailed evaluations. Between elevations 14'-1 ¾" and 17'-1 ¾", the concrete shelf will be subjected to lateral loads of about 170 kips from each support column which are directed towards the centerline of the reactor. Punching shear capacity of the concrete shelf at these locations exceeds the applied loads and thus the concrete shelf, along with the ¼" liner, is considered to resist the lateral loads from the six columns as a cylindrical ring subjected to an equivalent uniform pressure loading. The compressive stress in this cylindrical ring is found to be less than the Code allowable.

Biological shield wall and the foundation are considered adequate for the imposed loads by engineering judgment without performing detailed evaluations. The bearing pressure under the support columns and the contact area between the support girders and the concrete shelf are also found to be less than Code allowable.

The impact load on the pin connection at the bottom of the support columns is also found to be acceptable.

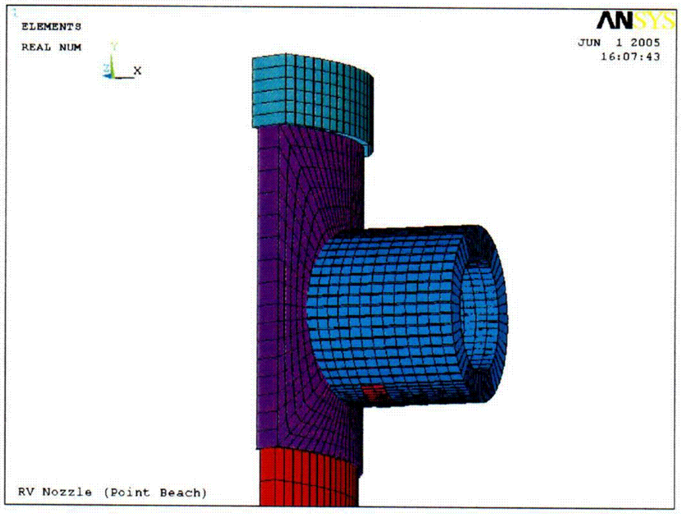


Figure 3.1 Reactor Inlet/Outlet Nozzle 60-degree Sector FEA Model

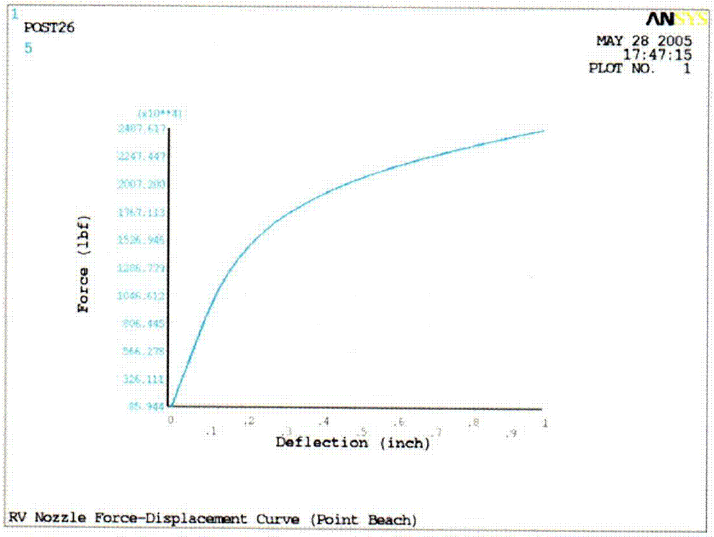


Figure 3.2 Reactor Inlet/Outlet Nozzle Load-Deflection Curve

C01

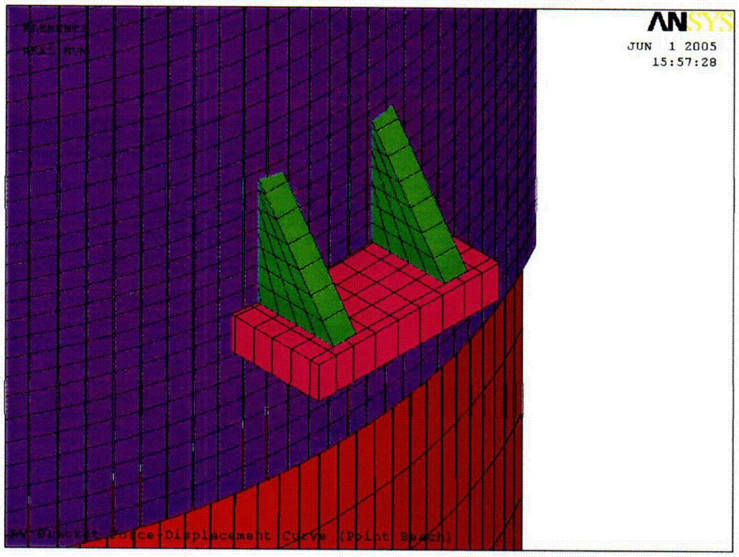


Figure 3.3 Reactor Support Bracket FEA Model

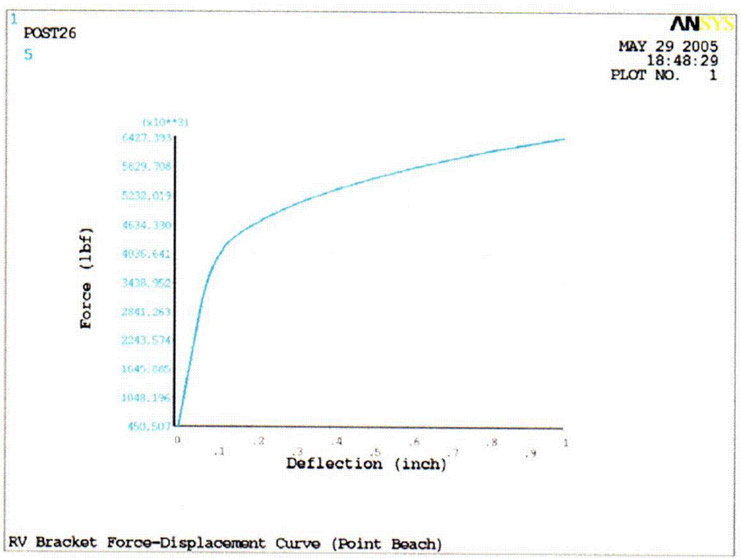


Figure 3.4 Reactor Support Bracket Load-Deflection Curve

C02

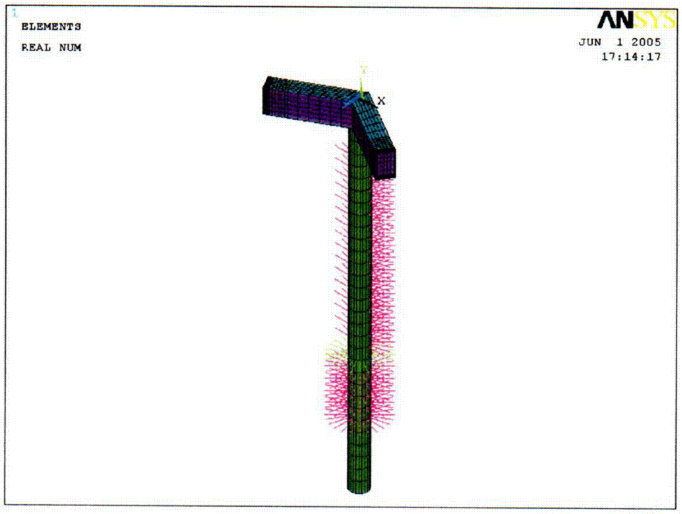


Figure 3.5 Girder Box Frame and Support Column FEA Model

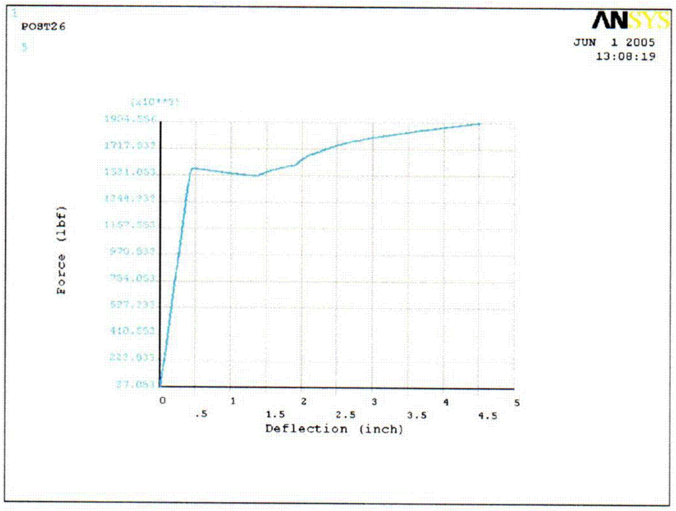


Figure 3.6 Girder Box Frame and Support Column Load-Deflection Curve

C03

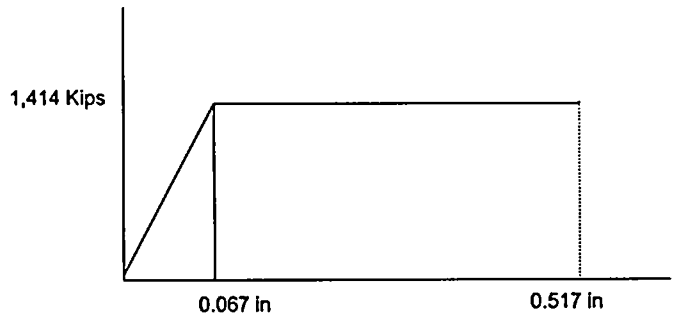


Figure 3.7 : Concrete Shelf Load Deflection Curve (Later)

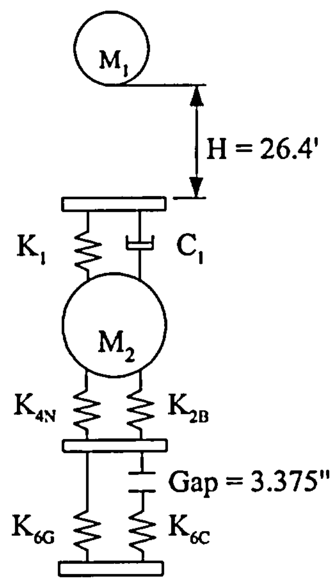


Figure 3.8 Dynamic Impact Model

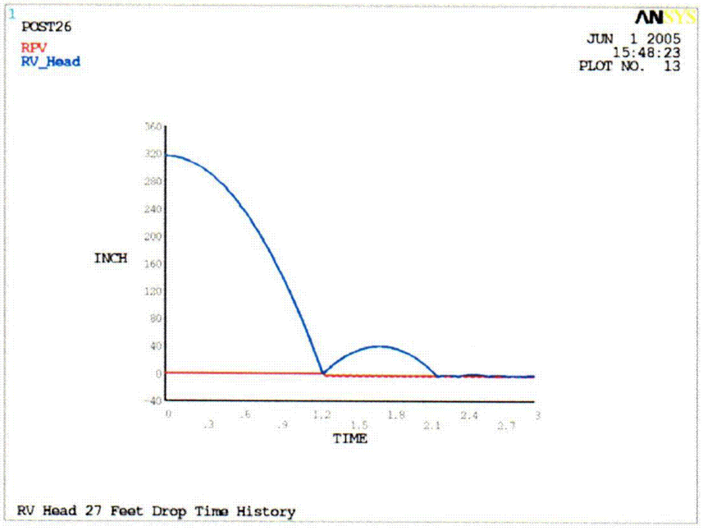


Figure 4.1 Dynamic Response of The Reactor Head Drop From a 26.4' Height

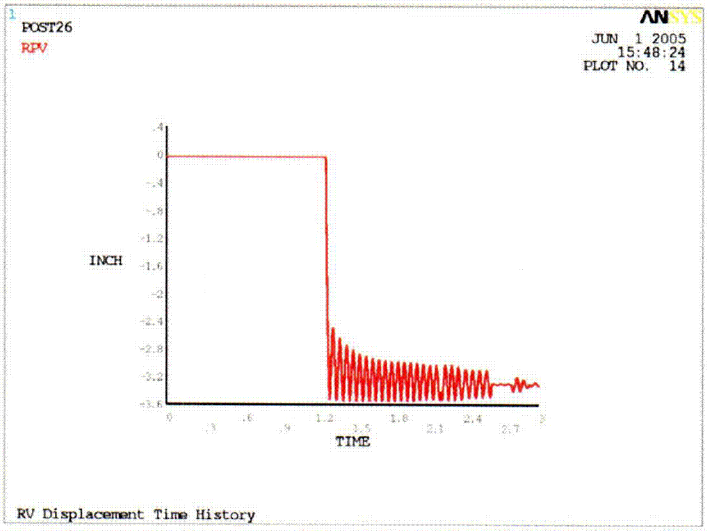


Figure 4.2 Dynamic Response of The Reactor Vessel Under the Head Drop Impact

C04

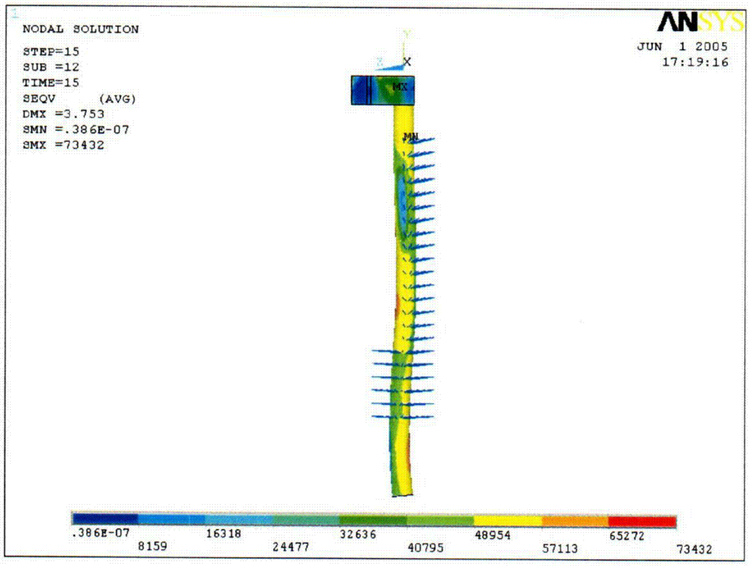


Figure 4.3 Girder Box Frame and Support Column Stress Under the Impact Load

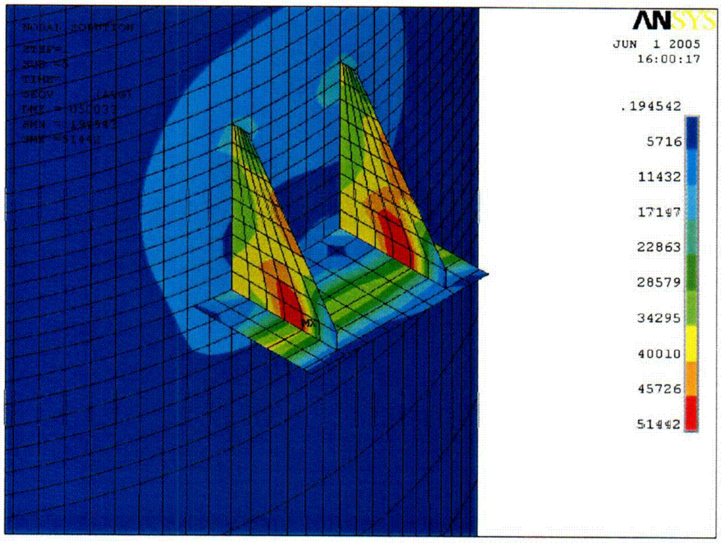


Figure 4.4 Reactor Vessel Support Bracket Stress Under the Impact Load

C05

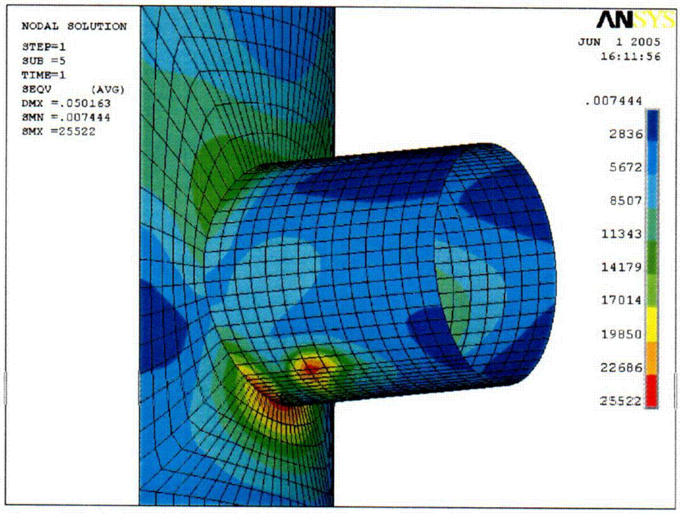


Figure 4.5 Reactor Vessel Inlet/Outlet Nozzle Stress Under the Impact Load

C06

ENCLOSURE 2

AUTOMATED ENGINEERING SERVICES CORPORATION REPORT,
"SUMMARY REPORT ON THE 3-D FINITE ELEMENT ANALYSIS
OF THE REACTOR VESSEL HEAD LOAD DROP FOR PBNP",
DATED JUNE 3, 2005

(9 pages follow)

AUTOMATED
ENGINEERING
SERVICES CORP.

387 Shuman Blvd., Suite 230W
Naperville, IL 60563
(630) 357-8880
Fax: (630) 357-4445
E-mail: avsetlur@aesengineering.com

June 3, 2005

Mr. Larry Peterson
Nuclear Management Company, LLC
Point Beach Nuclear Plant
6590 Nuclear Road
Two Rivers, WI 54241

Dear Mr. Peterson:

Subject: Summary Report on the 3-D Finite Element Analysis of the Reactor Vessel Head Load Drop for PBNP

References:
1. AES Letter Report on the Initial Reactor Vessel Head Load Drop Analysis for PBNP, dated May 29, 2005
2. AES Corp. Letter Report on the Initial Reactor Vessel Head Load Drop Analysis for PBNP, dated May 23, 2005

As stated in Reference 1, Automated Engineering Services Corp. (AES) in association with ANATECH Corp. (Bob Dunham) has performed the 3-D finite element analyses of the RV head drop using a 30-degree model of the head, vessel, nozzle, box girder and column A similar model for the bracket support is also evaluated.

This summary report describes the methodology and the results of two analyses

Evaluation Parameters

The following parameters are used in the load drop evaluation:

Drop Height in air:	26.4 feet
Replacement Head drop weight:	194 kips
Weight of RV above Vessel Support (W1):	137.6 kips
Weight of Vessel + Water - Head and W1:	570 kips
RV Dimensions and Wall Thickness:	From Westinghouse Drawings E 233-681 & -682, Rev. 0
Column Supports (6 total)	ASTM A-53, Grade B, 12" Sch. 120 Pipe, Pinned at Base, 20 feet long (actual column length =229")
Hexagonal Box Frame Support	From Bechtel Drawing C-2320, USS 'T1' material

Acceptance Criteria

The RV Head drop event is considered an accident event subjected to the guidelines given in Section 5.1 and Appendix A of NUREG-0612. Based on these guidelines, the following acceptance criteria are used in this evaluation:

a. Damage to the reactor vessel is limited so as not to result in water leakage that could uncover the fuel. This is interpreted to mean that permanent deformation of the vessel and its supports are acceptable provided that core-cooling capabilities are not lost.
b. The RV and its supports shall be capable of sustaining the vessel weight following the load drop accident.

c. The analysis should consider that all energy is absorbed by the structure and/or equipment that are impacted.
d. The RV supports including the nozzles should be evaluated for the transmitted impact loads.

Assumptions:

It is assumed that:

1. No energy is lost as a result of the impact. The entire kinetic energy of the falling weight is transferred to the target (RV and its support structure).
2. The base of the column supports are pinned and do not deflect.
3. The concrete biological shield wall enclosing the columns, nozzles and below the hexagonal support box frame do not participate in the load transfer. In other words, the columns are the only structural members that transfer the load to the foundation.
4. The load drop is vertical and symmetrical over the mating flange.
5. No credit is taken for the increase in stress limits due to the rate of loading. The dynamic increase factor for impact loading is conservatively neglected.
6. The column stress-strain relationship is linear up to the yield stress level followed by a plastic plateau up to 1% strain limit followed by a strain hardening linear stress-strain relationship to ultimate stress assumed at 10% strain. This stress-strain curve is conservative since the typical stress-strain curve for the A-53 carbon steel material is convex and envelops the assumed bilinear curve.
7. Although the guideline in Appendix A, Section 2 (6) of NUREG-0612 states that the RV head assembly should be considered rigid and not experience deformation during impact with other components or structures, this guideline is difficult to implement in an computer simulation. The RV head is an extremely rigid body that will not undergo any significant level of deformation. Therefore, the RV Head is treated as an elastic body with a thickness that results in the head weight of 194,000 lbs.

Methodology:

The load drop effect is simulated using the ABACUS/Standard computer software that has been validated and verified by ANATECH Corp., which is providing the finite element analysis to AES Corp.

Two separate finite element models are developed: the first for a section of the rector vessel that includes the nozzle support and the column and the second for the section containing the reactor bracket support and its column. Details are explained below:

Figure 1 shows a three-dimensional model of a 30 degree segment comprising the RV Head, the RV below the flange, the one-half of the nozzle, one-half of one side of the hexagonal support girder and one-half of the column pinned at its base. The 30-degree model was chosen to limit the size of the model while simulating the entire 360-degree vessel and supports through the use of proper boundary conditions at the cut locations. The model consists of 348 nodes with 281 3-D shell elements representing the head, vessel and the support structure.

For the reactor bracket support model, the finite element model remains the same except that the one-half of the nozzle model is replaced by one-half of the bracket. Note that the bracket consists of two 3" thick 18" deep and 14 5/8" wide triangular gusset plates with full penetration welds to the reactor vessel and to a bottom 4.5" thick horizontal plate that sits inside the support shoe that is supported by the hexagonal box girder which is finally support by the column. The brackets are SA516,

The boundary conditions applied to the model are as follows:

Circumferential displacement along the vertical boundary nodes – Restrained
Radial displacement along the vertical boundary nodes – Unrestrained except for the column base
Vertical displacement along Vertical boundary nodes – Unrestrained except for the column base
Proper rotational boundary conditions at the 0-degree and the 30-degree vertical cuts were imposed to simulate symmetry conditions.

The above boundary conditions assures symmetry of displacements within each of the 30 degree slices in plan while allowing unrestrained vertical movement of the head, vessel and its supports.

The material properties used for the various components of the model are taken from the CMTRs given in Attachment B and are as follows:

Reactor Vessel Head:	Elastic, E = 29 E6 psi;
Reactor Vessel:	Bi-linear, E= 29 E6; Yield = 35 ksi; Tensile Strength = 60 ksi at 10% strain
Nozzles:	Tri-linear, E= 29 E6; Yield = 35 ksi to 1% strain; Tensile = 80 ksi at 10% strain
Box Girder	Elastic-Plastic, E= 29 E6; Yield = 100 ksi since yield and tensile strength (105 ksi) are close
Column:	Tri-linear, E= 29 E6; Yield 42.8 ksi to 1% strain; Tensile = 78.9 ksi at 10% strain

At the mating flange interface an elastic contact element with compression-only property is introduced. This element ensures that the two flanges could separate during rebound of the head. The compression force is transmitted completely to the vessel wall and down through the support structure.

A time history analysis is performed with a time step of 0.1 millisecond. This time step is considered accurate to respond to frequencies up to 1000 Hz. From the results it is seen that the column vibrates in the axial direction at approximately 0.06 seconds or 16.7 Hz. This frequency compares well with the elastic first natural frequency of the columns of around 18 Hz. The high frequency content is around 300 Hz, which would be accurate responded to at a time step of 0.1 ms.

The analysis does not use any viscous damping assumptions. Hysteritic damping is inherently built in because of the bi-linear or tri-linear stress-strain curves for the material and because the column and parts of the shell experience plastic strains.

Results

Model with Nozzle Support

Figures 2 and 3 shows the variation of deflection with time at various locations of the nozzle supported FE model. The explanation of the tags used in this figure are as follows. Note that the 30 degrees and 0 degrees refer to the vertical slice locations at the 30 and 0 degree cuts:

h30bot135: head at 30 degrees, on the bottom, node 135
r30top130: reactor at 30 degrees, on the top, node 130
r30-124: reactor at 30 degrees, node 124
r30-115: reactor at 30 degrees, node 115
rnoztop161: reactor/nozzle top juncture at 30 degrees, node 161 (These first 5 are on a axial line down from the head at 30 degrees.)
r0-89: reactor at 0 degrees, node 89
r0-81: reactor at 0 degrees, node 81
gtop190: girder at 30 degrees, on the top, node 190

It is seen from this output that the reactor head is in contact for less than 5 milliseconds, which compares well with the axisymmetric results described in References 1 and 2. The first head bounce of the head of approximately 5.5 inches is also clearly seen in Figure 3. The reactor vessel and the columns settle approximately 3". Note that the displacements have a stable vibration and see only elastic ringing at the second impact. This shows that after the vessel and columns settle from the first impact, the subsequent loading behave elastically and the columns are able to support the dead weight load after impact.

Figures 4 and 5 show the column strains at various locations along circumference at the top. Bending of the column is seen from the strain variation around the circumference. Figure 5 clearly shows that the column is stable and is vibrating elastically after the first impact. The elastic ringing of the column following the second impact is clearly seen at approximately 0.35 seconds.

Model with Bracket Support

The results from the bracket model is very similar to that of the nozzle support model. Similar to the nozzle, the bracket is very stiff compared to the column stiffness. Therefore, the deformation and strains in the columns are not significantly influenced by the support arrangement. The deflections were slightly smaller than that for the nozzle model and the bounce effects were very nearly the same.

Therefore, the bracket support model responds very similar to the nozzle support model.

Strains and Plasticity in Other Parts of the Model

As expected, plastic strains are observed only in very localized areas near the vessel flange location and near the nozzles and supports. The strain levels ranged from elastic to approximately 0.9%. This localized amount of plasticity in the vessel is considered to be small and does not affect the dead weight carrying capability of the vessel or supports.

Stability of the Column Support

As discussed in Reference 1 and 2, the columns have remained stable and do not undergo buckling. The primary reasons for this is two-fold:

1. The impact load is extremely short lived (less than 5 millisecond) for the columns to react. Buckling is essentially a static phenomenon where the load is sustained for the secondary (P-Delta) effects to grow.
2. The column slenderness ratio is such that the column will go plastic before buckling occurs. In other words, the buckling loads are higher than the plastic load of the column.

Therefore, based on these extensive modeling with large deformation turned on and with the columns undergoing some amount of bending stresses near the box girders indicating that limited lateral displacements in the columns are present, the columns show that they will remain stable and will be able to sustain the dead weight of the vessel and its contents following the accidental load drop.

Conclusion

Based on the above, it is concluded that the reactor vessel and its supports will undergo around 3" of permanent settlement primarily in the column supports. The support structure will be able to sustain the dead weight of the vessel and the head following the load drop.

AES appreciates the opportunity to be of service to PBNP and NMC. We will be glad to respond to any questions or comments that you may have on this matter. Thank you.

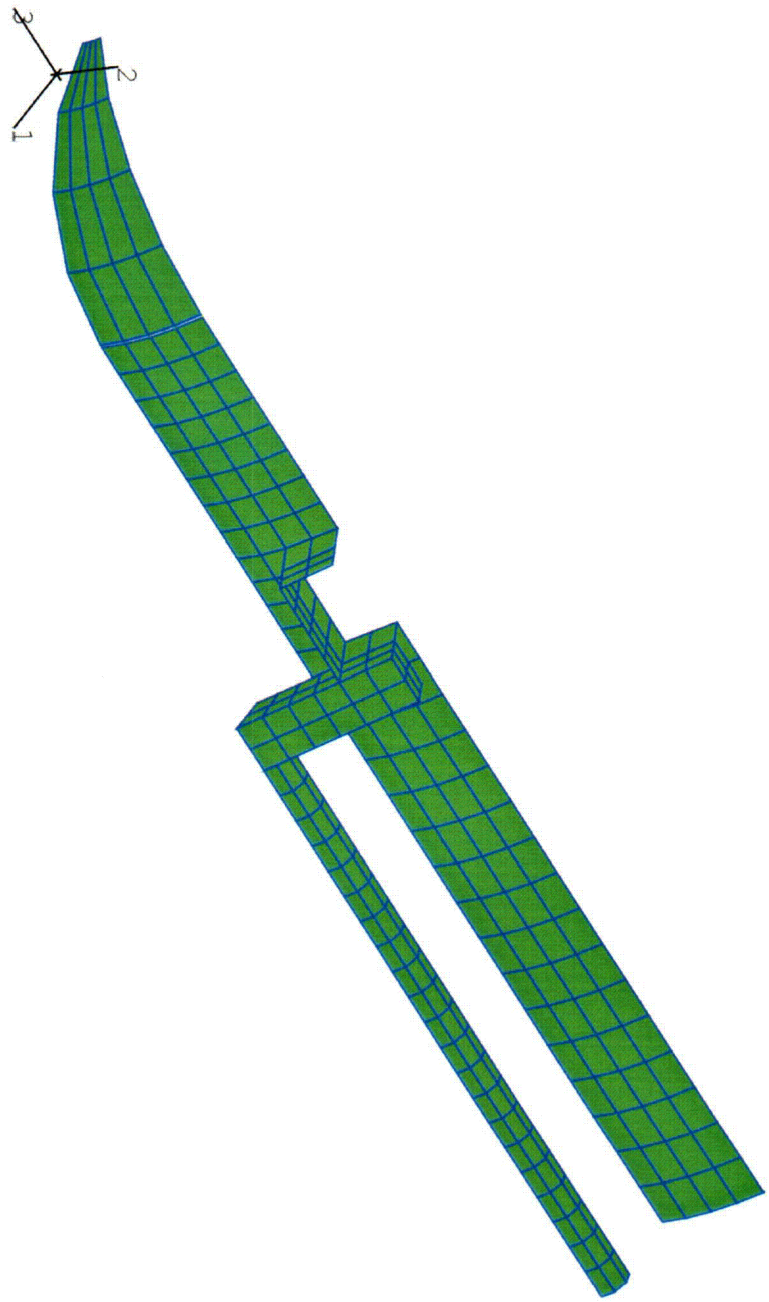
Sincerely,

A. V. Setlur

A.V. Setlur, Ph.D., P.E.
President

Cc: Joe McNamara, NMC
Robert Dunham, ANATECH Corp

Figure 1 – 3-D Model with Nozzle Support



C07

Figure 2: Displacement Vs. Time Following Impact

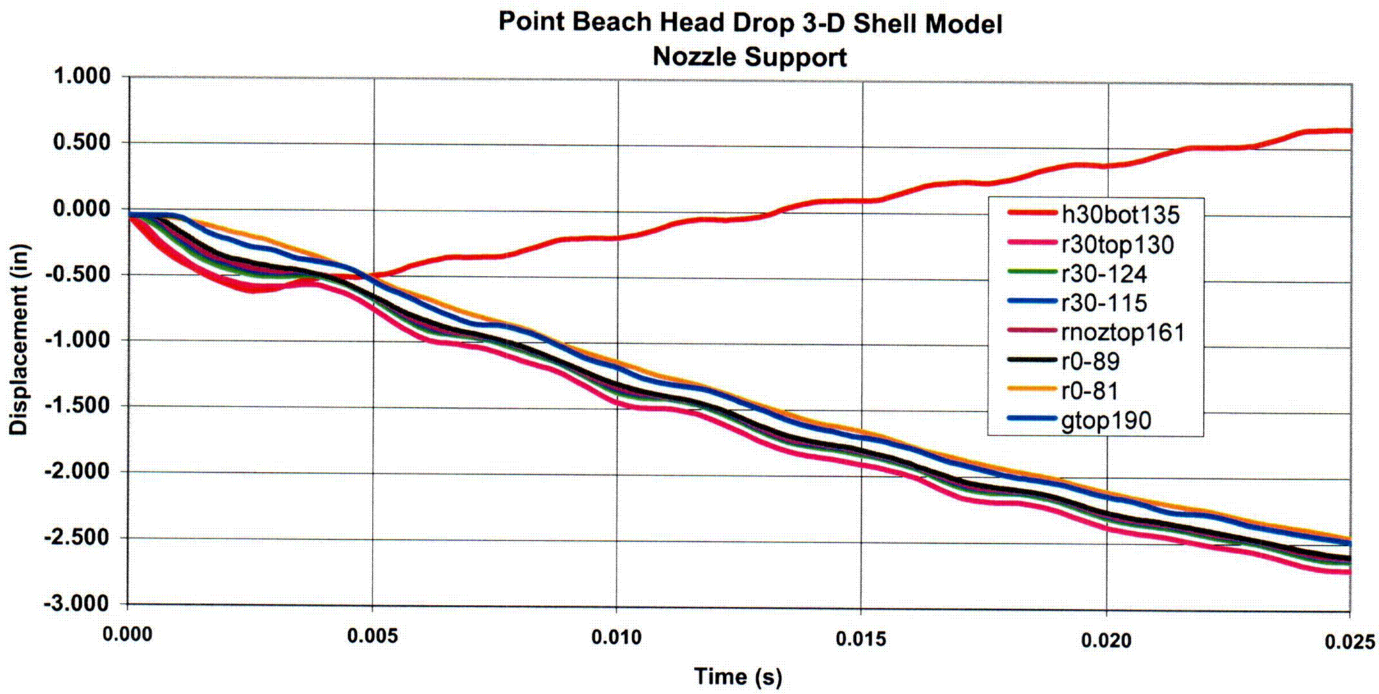
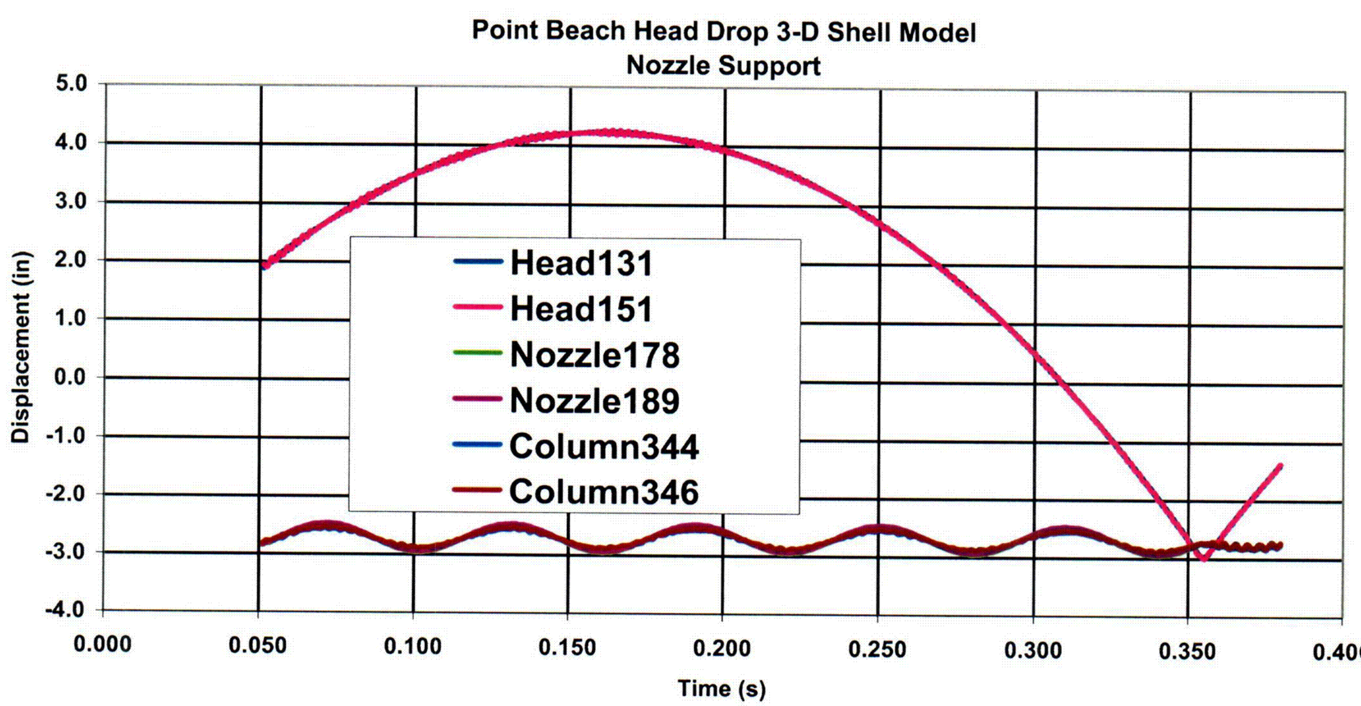


Figure 3: Displacement Vs. Time Following Impact (Extended Time to 0.4 secs)



C08

Figure 4 – Variation of Column Strains with Time

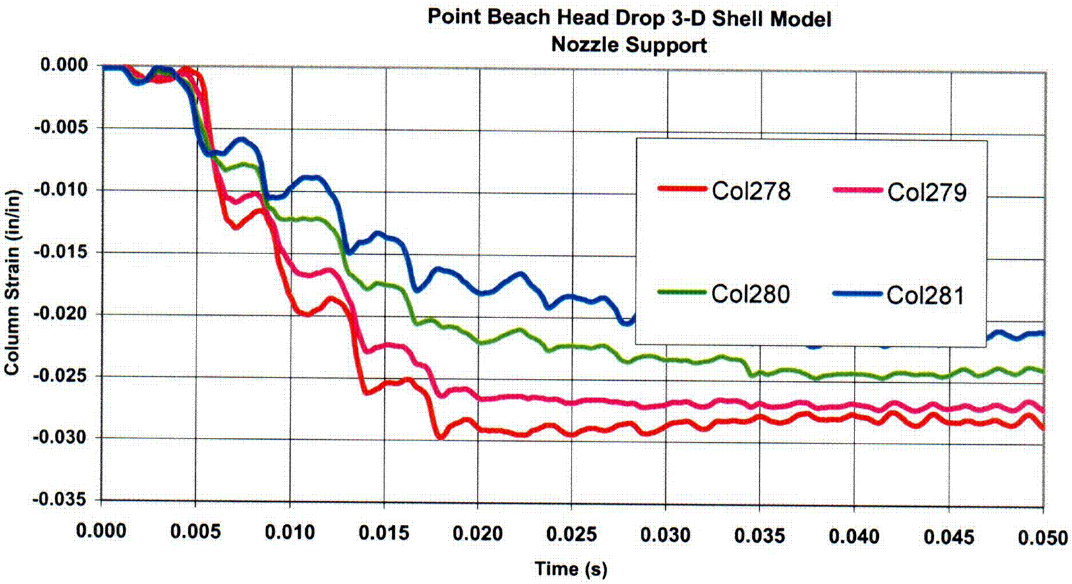
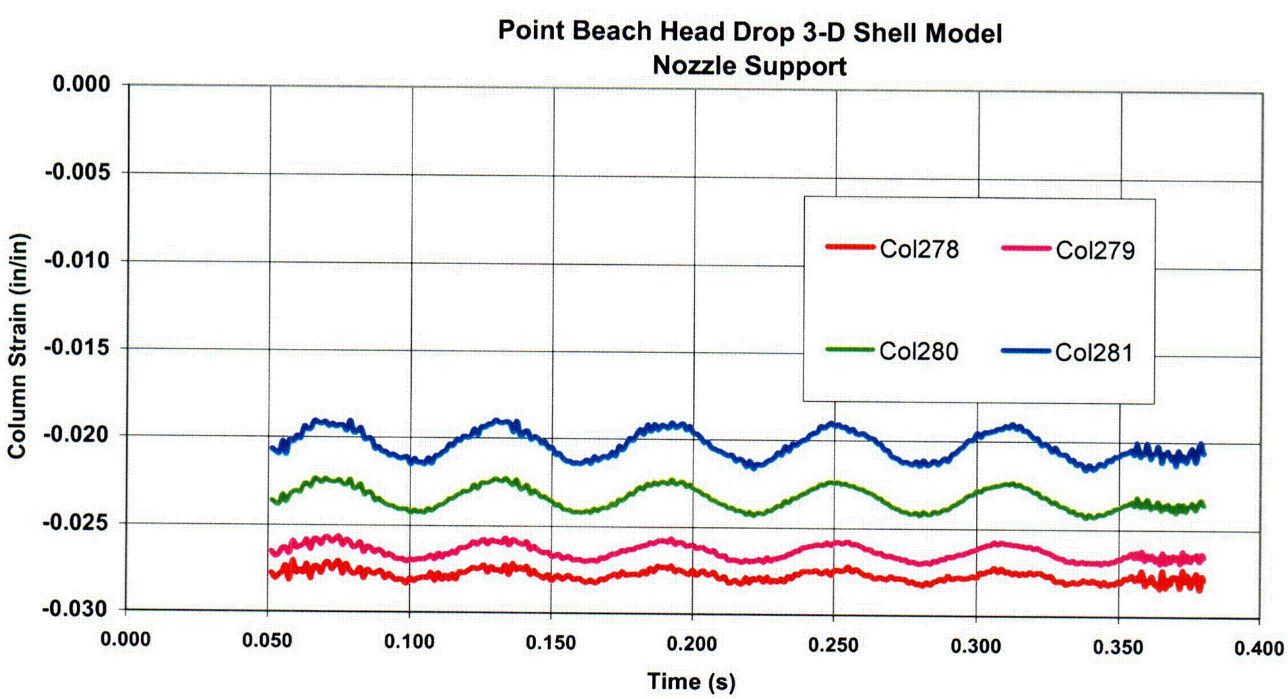


Figure 5 – Variation of Column Strains with Time (Extended Time)



C09

Figure 6 – Displacement Vs. Time Following Impact

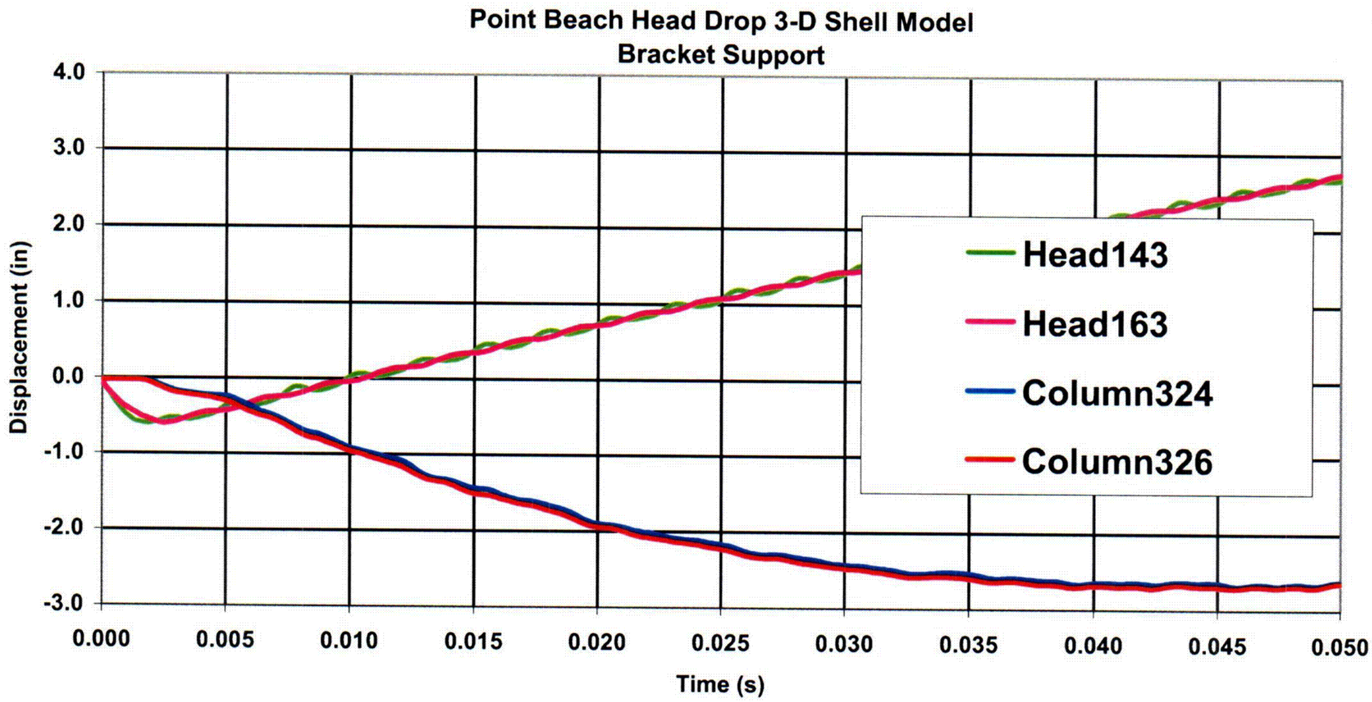
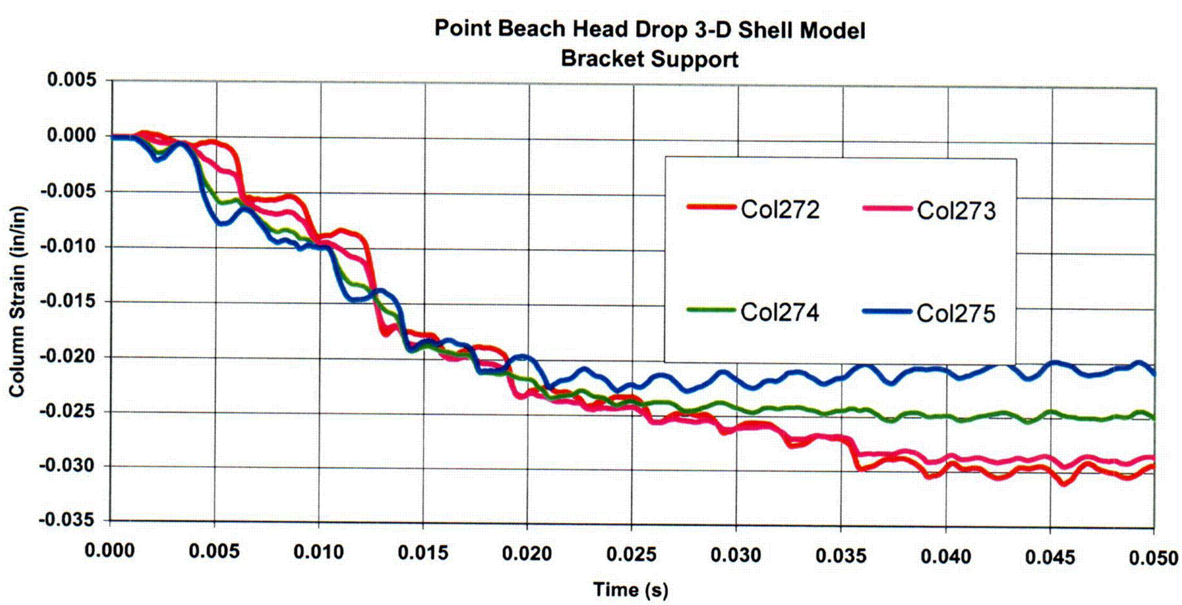


Figure 7 – Column Strain Vs. Time



C10