

ASME STRESS LINEARIZATION AND CLASSIFICATION – A DISCUSSION BASED ON A CASE STUDY

Carlos A. de J. Miranda¹, Altair A. Faloppa², Miguel Mattar Neto³, Gerson Fainer⁴

^{1,2,3,4} Instituto de Pesquisas Energéticas e Nucleares (IPEN / CNEN - SP)
Av. Professor Lineu Prestes 2242
05508-000 São Paulo, SP
cmiranda@ipen.br¹, afaloppa@ipen.br², mmattar@ipen.br³, gfainer@ipen.br⁴

ABSTRACT

The ASME code, specially in its Nuclear Division (*Subsection NB – Class I Components*), gives some recommendations to the structural analyst on how to perform the verifications required to prove the design as good as the by-analysis prevented failures modes. Each of these failure modes has specific stress limits which are established based on simple but conservative hypothesis like the material perfectly plastic behavior and the shell theory with its typical membrane and bending stresses with linear distribution along the thickness. Other detail to keep in mind is the code distinction between primary and secondary stresses (respectively, stress that came due to equilibrium and due to displacement compatibility). In general, the numerical models used in the analyses are developed with plane or 3D solid elements and due this fact no direct comparison with the code limits can be done and, besides that, the programs do not distinguish between primary and secondary stresses. Mostly, the later are produced due to the temperature variation but they also appear near discontinuities. Sometimes, this classification is not so clear or direct. To perform the required ASME Code verifications the analyst should obtain the membrane and bending stresses from the plane or 3-D model which is called stress linearization and, also, should classify them as primary and secondary. (The excess between the maximum stress at a point and the sum of these linearized values is called peak stress and is included in the fatigue verification.) This task, most of the time is not a simple one due to the nature of the involved load and/or the complex geometry under analysis. In fact, there are several studies discussing on how to perform these stress classification and linearization. The present paper shows a discussion on how to perform these verifications based on a generic geometry found in many plants, from petrochemical to nuclear, which emphasizes some of these issues.

1. INTRODUCTION

The ASME code, particularly in its Section III [1], the nuclear one, has specific requirements on how to assess the results from the stress analyses to make the necessary verifications to avoid failure. These requirements apply to those equipments (valves, pumps, vessels, piping, etc.) which are part of the pressure boundary.

The components, as well as the piping in a nuclear power plant, should be classified accordingly to its nuclear safety classification, as Class 1, Class 2, Class 3 or Class NN (non nuclear). The project of nuclear equipment classified as Class 1, 2 or 3 should follows the recommendations given in the sub-sections NB, NC and ND, respectively. The present work deals with the structural and stress analysis of Class 1 equipment which structural project should be verified “by analysis” instead of “by rules”. The later is the ‘conventional’ way to verify a project before the massive use of the numerical methods as the finite element method. Generally speaking, a Class 1 piping analysis should be performed and its results

analyzed accordingly to the item NB 3600, Section III, of the ASME Code while the analyses of Class 1 equipment as the Vessels in general (Reactor Pressure Vessel, Steam Generator, Pressurizer, et.) should follow the item NB-3200 and NB-3300.

Usually, to analyze equipment one uses a model made basically by shell and solid elements (2D, 3D or axi-symmetric ones). To analyze a piping, besides the general straight and the curved pipe elements, the analyst has to model some standard parts or fitting as valves, short radius and long radius curves, Tees, etc. Supports and anchors are also used. Standard curves and Tees, for instance, have specific stress indices (that behaves like stress concentration factors) which are experimentally determined as function of the fatigue life reduction with respect to the straight tube fatigue life. These factors are given by the applicable code, specifically, in the item NB-3600 for Class 1 nuclear piping.

In some plants the space restriction imposes a compact lay-out with the use of non-standard items. For those special piping items with non-standardized geometry the piping analysis can not capture the actual stress state and, therefore, a specific analysis should be performed using the requirements of the item NB-3200/NB-3300. In this situation, usually the analysis is performed in two steps. Firstly, the piping is analyzed (and modeled) as usual where the special item is modeled by rigid straight tubes (with an equivalent cross section and density to introduce its mass in the model). All model results, except by the special item, should attend the NB-3600 limits. In the second step the special item is modeled as equipment, with shell and/or solid elements. Forces and moments from the piping in its interface should be applied as well as all other loads like pressure and temperature. If the boundary conditions are not so obvious, the whole piping can be introduced in the model as a super-element. At the interface solid-tube a rigid region can be created connecting the tube central node with all nodes in the respective 3D interface. The results (stresses in general) from the special item, but not the piping super-element results, should be verified against the NB-3200 limits.

This is done because the piping programs have routines to verify the results while the general purpose programs do not have these routines. In the nuclear area one of the most used program for piping analysis is the PipeStress [6] while the ANSYS program [4] is largely used for equipment shell and solid modeling. The ANSYS program has piping elements but it does not have the post-processing routine to perform the required NB-3600 verifications. And it should be very expensive and time consuming to develop such tool for a given project. The results verification against the NB-3200/ NB-3300 limits is not always a direct step. It often needs an intermediate step called stress linearization in a given section once most of the limits are in the average stress (membrane stress) or they are in the linear part of the stress distribution (membrane + bending stress). The total stress is used in the fatigue verification.

Another care that should be taken when using the ASME NB code in the verification of the results from the stress analysis is the stress classification as Primary and Secondary. The former is related with the equilibrium equations and their limits are mandatory. Typically, they are from the mechanical loads. If they are not attended the failure is immediate. The secondary stresses are related with the compatibility equations. Typically, they are from the thermal loads and the failure is not immediate and it is, usually, related with the load cycling.

These steps, the stress classification and the stress linearization, are not straightforward ones and needs some 'engineering' judgment to choose the right section to evaluate the stresses in discontinuities. Often it is necessary to choose several sections looking for that most stressed

one. And more often, some sections are invalid to perform the classification and linearization procedure as shown by researchers.

The scope of this work is to describe the second step of the above procedure and to discuss how to perform these stress verifications using a generic geometry found in many plants, from petrochemical to nuclear, which emphasizes some of these issues that arises from the stress classification and linearization in discontinuities, which are common in the nuclear area. The chosen geometry is a four-way Wye junction [3], connecting, in the inlet side, three piping and one piping from the outlet side. All this piping is supposed to be Class 1 from a nuclear plant. All adopted material properties and loads, as well as, dimensions and lay-out, are typical for a small nuclear plant.

1.1. Modeled Geometry

The overall lay-out is presented in Fig. 1. It shows the piping parts (modeled as tube elements: straight and curved ones) and the junction (modeled as rigid tube elements). The Wye junction itself is presented in Fig. 2. In the inlet side its main dimensions are: external/internal diameter 219/173 mm. In the outlet side the dimensions are: external/internal 273 / 216 mm. The junction is about 800 mm high. In both sides the common dimensions are: straight segment length 160 mm, fillet radius 20 mm.

In the analyzed junction it is supposed to act only the internal pressure and it is isolated from the piping. This was done to simplify the analysis procedure aiming to strengthening the discussion on how to classify and linearize the stresses in the discontinuities. With this hypothesis and considering the double geometric symmetry it is possible to model only $\frac{1}{4}$ of the junction.

2. FINITE ELEMENT MODEL

Mesh. The finite element model was developed, and the stress analysis was performed, using the ANSYS APDL program [4] from a solid model developed in the SolidWorks program [5]. This model was imported to the ANSYS using the parasolid format (extension “.x_t”). The junction was modeled using the 3D solid isoparametric element named SOLID95 in the ANSYS element library [4]. This element has quadratic interpolation with three nodes in each border and it has 20 nodes in its basic form and can be degenerated to a prism or a tetrahedron. If the piping was also to be modeled, the PIPE16 (straight) and the PIPE18 (curved) elements should be used.

The mesh was generated defining, initially, an element size (13mm) and, for some lines, chosen by visual inspection, a specific number of divisions were defined. This approach was done to assure to have, always, at least 5 nodes in a given line across the thickness (this is represented by at least two elements in the thickness). The generated mesh can be seen in Fig. 3.

Material properties. As material properties it was adopted: Young modulus $E = 180 \cdot 10^9$ N/m², Poisson’s ratio $\nu = 0.3$, mass density $\gamma = 8500$ Kg/m³. These values were taken from [2] supposing an ASME material used for nuclear applications at a typical design temperature $T_{des} = 350$ °C. At this temperature the reference stress and the yield stress are, respectively, $S_m = 110,3$ MPa and $S_y = 122,7$ MPa.

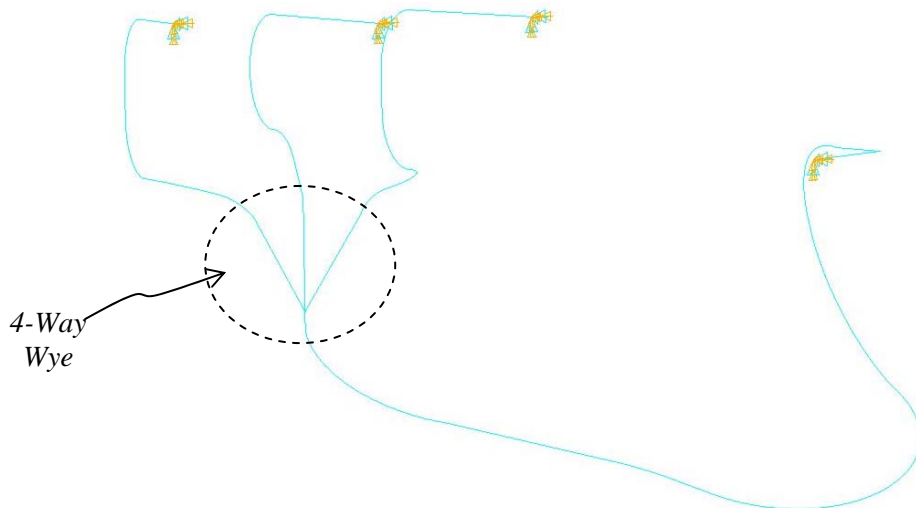


Figure 1. Lay-out showing the four-way Wye junction as part of a postulated piping.

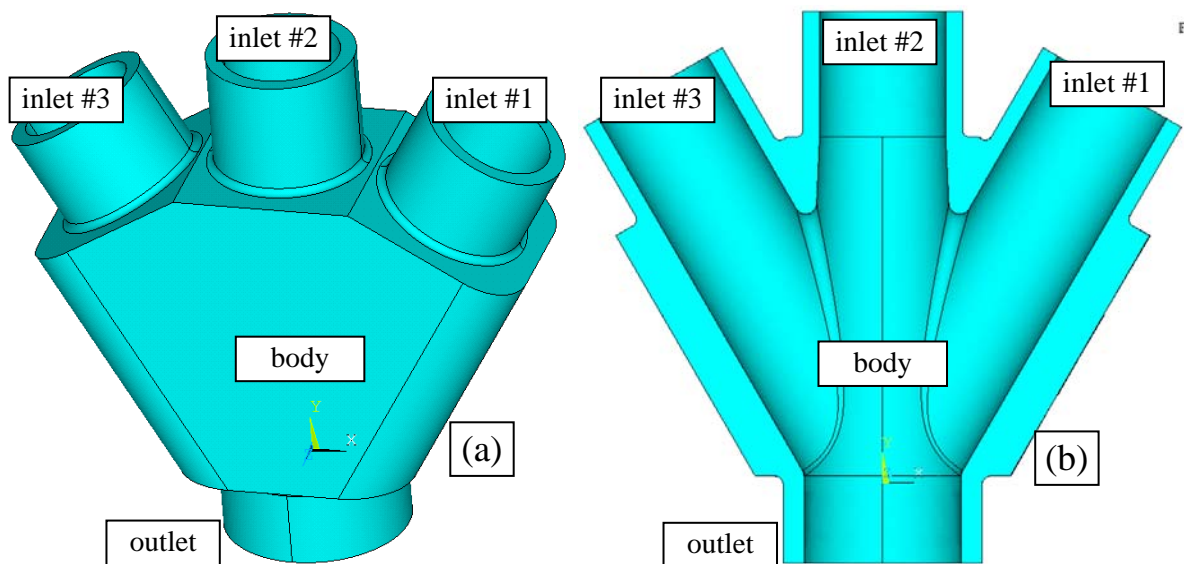


Figure 2. Four-way Wye: (a) solid model, (b) internal view

If other non-mechanical loads are concerned, as thermal ones, a thermal expansion coefficient $\alpha = 17.6 \cdot 10^{-6} \text{ mm/mm}^\circ\text{C}$ should be used to evaluate the thermal strain and stresses. (In an actual situation there are other loads as dead weight, earthquake, thermal transients, etc. To consider the dead weight for instance, the water and thermal isolation should be considered besides the steel density. To do so, an equivalent density should be calculated and it can reach values around $\gamma_{\text{eq}} = 11000 \text{ Kg/m}^3$.)

Loads. As mentioned before, for this analysis, the only load considered was the internal pressure with a typical value, $P_{\text{int}} = 15 \text{ MPa}$ applied on all internal surfaces. Besides this load, and to be coherent with the physical and mechanical behavior of a vessel under internal pressure, some “closing forces” should be applied at the openings which work like nozzles. These ‘forces’ were applied as longitudinal stresses which, for this particular situation, reach

$Pr/(2.t) \approx 32$ MPa in all the four openings (P is the internal pressure, r and t are the mean radius and the pipe thickness at each opening).

Boundary conditions. Considering the assumed double symmetry, once the only applied load is the internal pressure, the imposed displacements were those related with the symmetries in the nodes on the symmetry planes (at $X = 0$ and at $Y = 0$). The nodes on the model bottom part were restrained in the Z direction.

Model continuity and compatibility. In an actual situation, the piping would be connected to this 3D model as depicted in Fig. 1 and the closing forces would be still necessary to make the model continuity and compatibility (the piping elements consider these forces in their formulation but not the 3D elements). And the imposed displacements would not be applied as described above and depicted in Fig. 3 but, instead, they would be applied in the piping extremities (as shown in Fig. 1). Special care should be taken at the interface solid-tube: besides the closing forces, a rigid region should be defined connecting the piping node (*'master'*) with all nodes in the respective 3D interface (*'slave'*). This is done using the ANSYS CERIG command.

Other loads. If there are other loads as dead weight and/or earthquake the symmetry can not be adopted and the whole model (pipe + 3D elements) should be considered. The dead weight is automatically considered by the program once a material density is defined and the gravity acceleration is applied to the model. The seismic loads could be applied in the junction model as forces and moments that came from the piping analyses due to the response spectrum analysis, for instance. These forces and moments should be the maximum values obtained in each direction. In this way, the seismic load parcel due to the junction own weight is not considered.

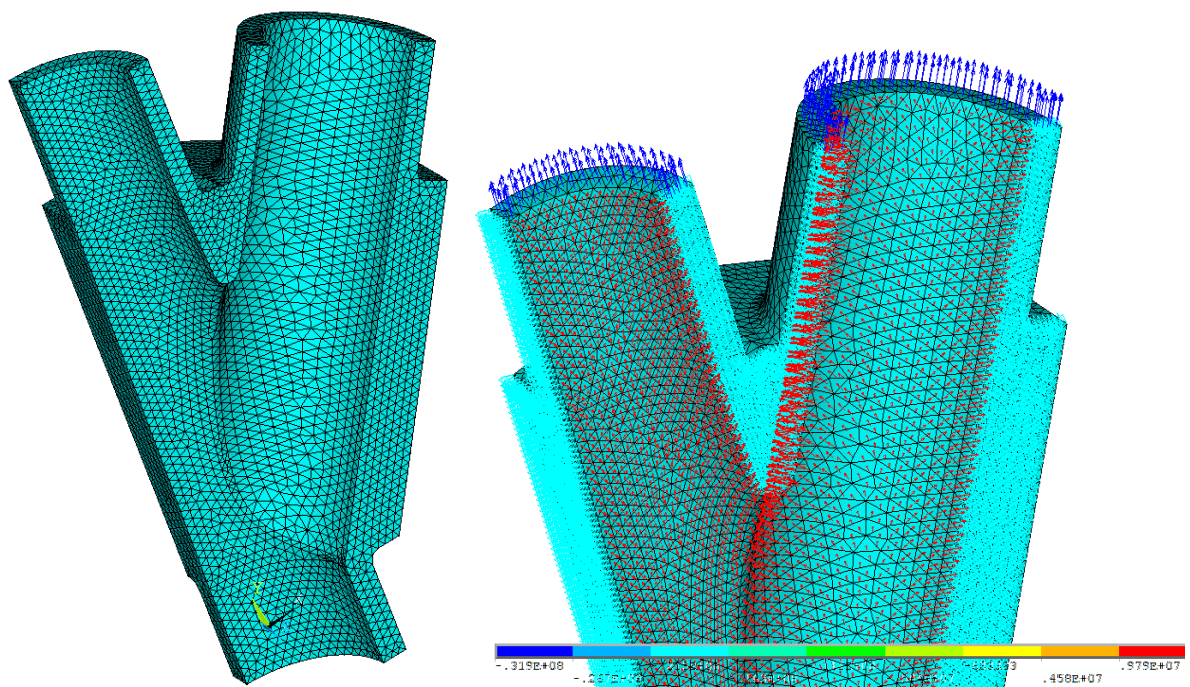


Figure 3. $1/4$ model and mesh adopted and detail with the applied pressure, closing forces and symmetry conditions in the model upper part

This is a reasonable hypothesis once in the piping analysis seismic loads this weight was already considered once the junction was modeled as rigid element with its own mass. This hypothesis is more exact if the junction, considered isolated, has its first frequency ≥ 33 Hz. (To superimposed the seismic loads, that came from a response spectra analysis, with the other loads two load steps are necessary: in the first all the seismic loads are considered with the positive sign and in the second the seismic loads are considered with the negative sign.)

3. STRESS CLASSIFICATION AND LINEARIZATION

The stress classification is done to identify, basically, the “Primary” (P) and the “Secondary” (Q) stresses. The former are directly related with the equilibrium equations while the later are related with the compatibility equations. So, in general, they came respectively from mechanical and thermal loadings. But in a structural discontinuity there are a stress increasing due the need of compatibility between the connected parts. So, even for mechanical loads there are secondary stresses in a given discontinuity.

As the ASME limits [1] are developed aiming to prevent some typical failure modes besides the Primary and Secondary classification, the stresses should be linearized to obtain the generalized (P_m) or localized (P_L) membrane component, the bending (P_b) and the Peak (F) stress. This linearization should be done along a cross section of the piping or equipment but in the discontinuities this section is not so clear. In such a case the stress linearization is done along a line, the SCL – Stress Classification Line. This problem does not exist in beam models neither in shell models once these models ‘naturally’ give the stresses separated in membrane and bending components. The use of stress intensification indexes should be considered. The problem arises strongly in 3D models which are very common nowadays.

Albuquerque [7] discusses in detail the procedure and equations related with this linearization procedure considering several situations/sections: plane model (2D), axisymmetric model, general 3D model and if it is performed on a section, on a line (SCL). The work in ref. [7] is based on recommendations given by researchers, mainly Hollinger & Hechmer (H&H), after numerical modeling/analysis performed using several geometries. Their work started in 1985 and their findings had being consolidated in [8, 9, 10].

The average value acting on the whole section/line that is equivalent to the net force acting in the section due to the actual stress distribution will be classified as P_m or P_L depending on the distance of the section from the discontinuity: P_m for those far sections and P_L otherwise. This P_L classification is justified because there is a secondary ‘aspect’ in this stress near a discontinuity even if it comes from a mechanical load. Eq. (1) defines when a section is near a discontinuity (d is the section distance from the discontinuity, r_m is the average radius and t the tube thickness).

$$d \leq 2.0 \sqrt{r_m t} \quad (1)$$

The maximum value of the linear stress distribution which produces a net bending moment equivalent to the moment produced by the actual stress distribution is called ‘bending stress’ P_b. For mechanical loads, if the section is near a discontinuity this stress component is classified as secondary, ‘Q’ (once it is due to the deformation compatibility between the connected parts).

The difference between the actual stress distribution and the sum of the average and linear (membrane + bending) stress distributions give an auto-equilibrated stress distribution which produces neither a net force nor a net moment in the section - which maximum is the so called “Peak” (F) stress. In a given section, the maximum stress occurs, in general, in one of its surfaces (internal or external) and it is the sum P_m (or P_L) + P_b + F.

The stress classification adopted in the present work, for those sections or SCL where the stresses were verified, follows the ASME NB Table-3217-1d [1] and to chose the SCL the recommendations from H&H in [10] are followed, specially its appendix V from which the Fig. 4 was adapted.

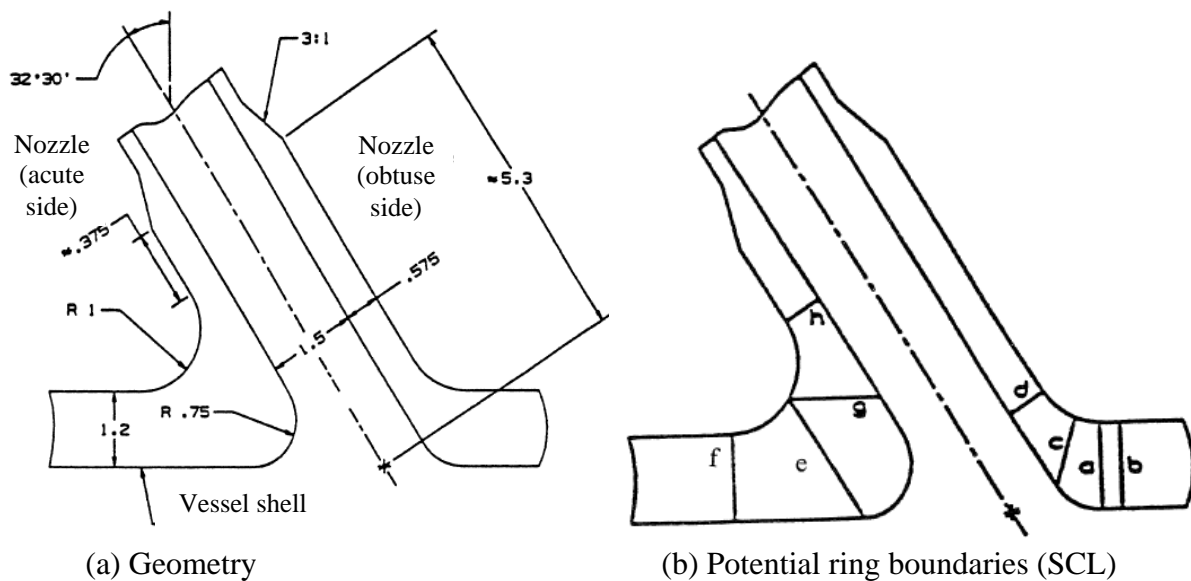


Figure 4. Nozzle to cylinder analyzed geometry in the WRC 429 (adapted from [10])

Taking the geometric configuration depicted in Fig 4 one can notice that the ring boundaries vary around the circumference of the nozzle-shell junction. In [10] H&H analyzed several options for these boundaries definitions based on two approaches: one minimizing the size of the ring and other maximizing as it can be seen in Fig. 4(b). To evaluate each possible boundary, they used the component stress distributions along each line/boundary to determine its consistency with the shell stress distributions which means, the validity of the SCL.

H&H evaluated the SCL in the acute side and in the obtuse side based on (a) SI (stress intensity) and principal stresses and (b) component stress distribution along the lines. As an example of the later, the distribution of the hoop stress should be nearly linear and the bending should be low (in special for those lines that are radial to the vessel), the shear stress should show low values and a parabolic distribution, and the radial stress should be also nearly linear with low values (if the line is perpendicular to the surface, the stress inside the internal surface should be compressive and equal to the internal pressure and in the external surface it should be zero).

In the present work the three tubes in the junction have the same diameter and thickness (Fig.

2) while the geometry analyzed by H&H represents the junction of a small nozzle at an angle to the shell axis (Fig. 4). However, the late represents well the difficulties to establish the SCL to perform the Code verifications and, so, in the present work the H&H main conclusions will be used as a guide to establish the SCL in the four-way Wye junction, mainly in the acute sides.

Considering the main piping dimensions in the analyzed geometry, the minimum distance to consider a given section (or SCL) far from a discontinuity is $\approx 95\text{mm}$ for the inlets and it is $\approx 118\text{mm}$ for the outlet side. These values are adopted also for the 'body' part of the Wye (see Fig. 2). So, only in the middle of the inlet #2 (the central one) where the arriving piping is considered to be aligned with the Wye piping segment (see Fig. 1) there could be a section or SCL where the stresses should be classified as P_m . In all other section the averaged stress should be classified as P_L once the section is near a discontinuity (and the bending linear component should be classified as Q (secondary), see Table-3217-1d [1]).

Limits and Verifications. The allowable values for each type of stress (P_m or P_L , $P_L \pm P_b$, $P+Q$, $P_L \pm P_b + Q + F$) are derived from the basic allowable - S_m - at the working temperature. To stay within the scope of this work and to be coherent with the applied load (pressure), only the primary stress (P_m , P_L and $P_L \pm P_b$) will be verified. So, accordingly to Fig. NB-3221-1 and NB-3222-1 of the ASME NB [1] the due verifications and their limits are:

$$P_m \leq S_m = 110.0 \text{ MPa}; P_L \leq 1,5S_m = 165.0 \text{ MPa}; P_L \pm P_b \leq 1.5S_m = 165.0 \text{ MPa}.$$

The indicated limits are for the so-called Design Condition. When the Operational Conditions are verified the pressure and temperature are lower, but the earthquake should be considered, the limits are slightly different as well as the S_m value (which depends on the temperature). For more information and details, see NB 3223.a.1 [1].

Sections to Linearize, Classify and Verify the Stresses. Ten sections or SCL were chosen to linearize and classify the stresses, to cover all critical parts and regions in the analyzed geometry, aiming to verify them against the Code limits. These sections are shown in Fig. 6 where they are named PATH_xx where 'xx' is the section number.

From above, the section #01 (e.g., PATH_01 in Fig. 6) is the only one which averaged stress should be classified as P_m (general membrane stress) while for all other sections the average stress should be classified as P_L (localized membrane). This is a conservative approach once this stress is affected by the discontinuity and there is a parcel on it that came from the deformation compatibility. Other conservative approach, widely used in 3D solid models, is the stress linearization along a line through the thickness instead of their linearization in a section. The sections 06 and 07 were chosen to comply with the criteria in the Appendix V of the WRC 429 Bulletin [10]. It should be pointed that the sections 02 to 10 are not suitable to stress linearization, accordingly the appendix V of ref. [10], once they are directly on the transitions. Again, following [10], if one wants to verify the stresses in the transition, the sections 05 and 06 are better than the section 07 to perform the stress linearization.

The bending stress will be classified as Secondary (Q) and it will not be discussed in this work. Their verification should also consider the stresses from variable loads and loads that produce secondary stress (as the thermal ones). That means, including fatigue verification, other set of analysis which is out of this work scope.

4. ANALYSIS RESULTS

In a point under a stress tensor with six different components the Tresca equivalent stress, SI – Stress Intensity, is defined as in Eq. (2). The Tresca stress (and not the von Mises one) is adopted by the ASME Code Section III Sub-Section NB [1]. Fig. 5 shows the ¼ model expanded to form ½ of the analyzed four-way Wye under internal pressure while Fig. 6 shows the SI (Stress Intensity) distribution in the upper region of the Wye with the superimposed sections. In both Figures, the stress scales (in N/m²) were manually defined. (In actual situations the stress field will not be symmetrical due to the other loads and the asymmetry introduced by the piping which will not allow a ¼ or even a ½ model.)

$$SI = \max(|\sigma_1 - \sigma_2|, |(\sigma_2 - \sigma_3)|, |(\sigma_1 - \sigma_3)|) \quad (2)$$

Linearized Stresses. Figures 7 and 8 show the typical graphic form to present the linearization results for sections 1 to 8 in terms of the SI stress. In its post-processor module the ANSYS program can linearize the stresses along a given section defined by two points – the SCL. It linearizes all six stress components (SX, SY, SZ, SXY, SYZ, SXZ). Also, the Tresca (SINT) and von Mises (SEQV) equivalent stresses can be presented. To perform the linearization the program considers 50 internal points along the SCL. With this procedure, the Membrane (average), the Bending (linear), the Membrane ± Bending, the Peak stress, besides the total stress, are calculated and presented in table as well in graphical form.

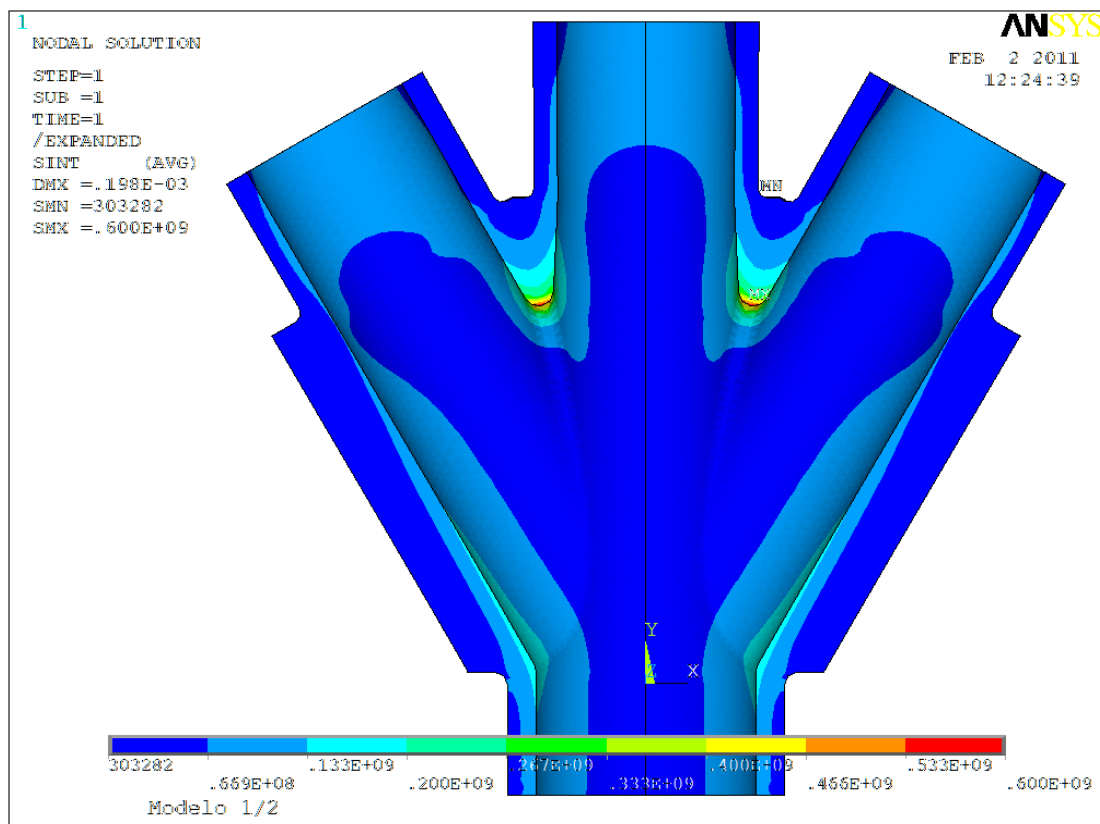


Figure 5. SI stresses (N/m²) in the ¼ 3D solid model expanded to one-half.

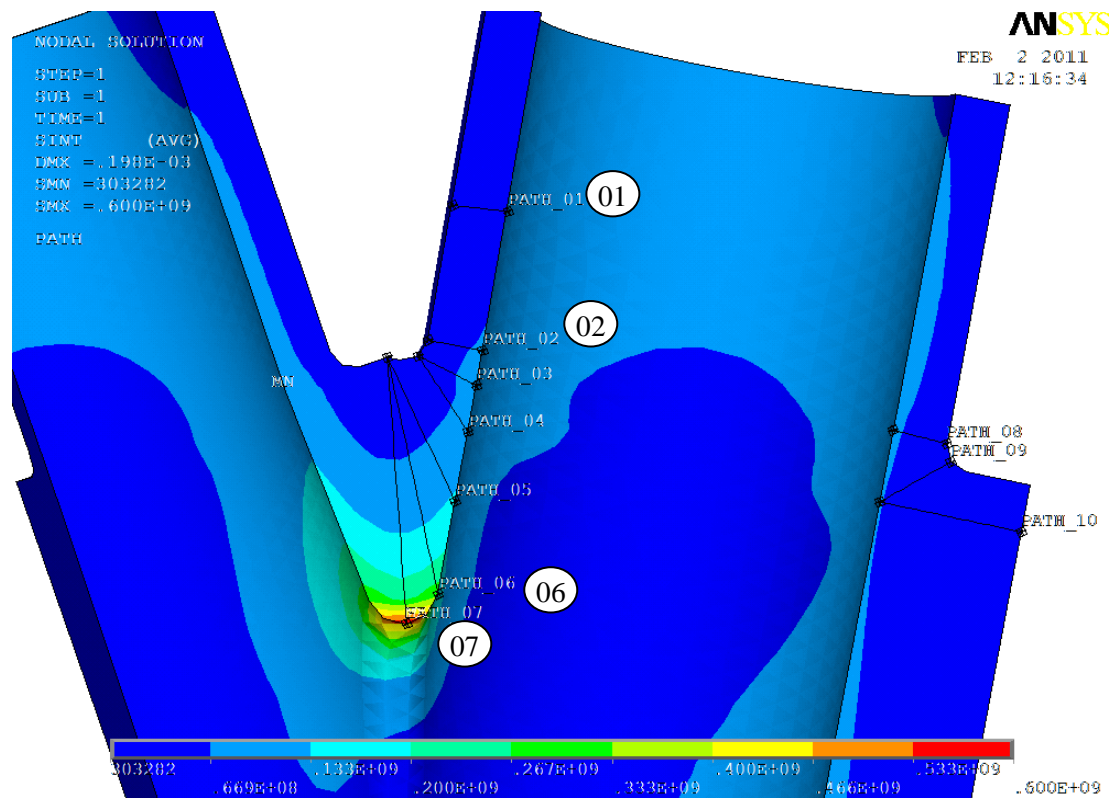


Figure 6. Stress Intensity (SI) distribution (N/m^2) in the $\frac{1}{4}$ model upper part

One can notice, looking at the results for sections 1 to 7, presented in Fig. 7, how the stress distribution becomes irregular and far from the theoretical linear distribution as one approaches and go ‘inside’ the geometry transition, as discussed in [7 - 10].

Primary Stresses Verification. Table 1 shows the verifications performed in every one of the 10 defined sections in terms of P_m , P_L and $P_L \pm P_b$ (Primary stress) as well as their limits and the ratio (in percentage) between the stress and its limit. All values are within the respective Section III Sub-Section NB ASME Code [1] limits.

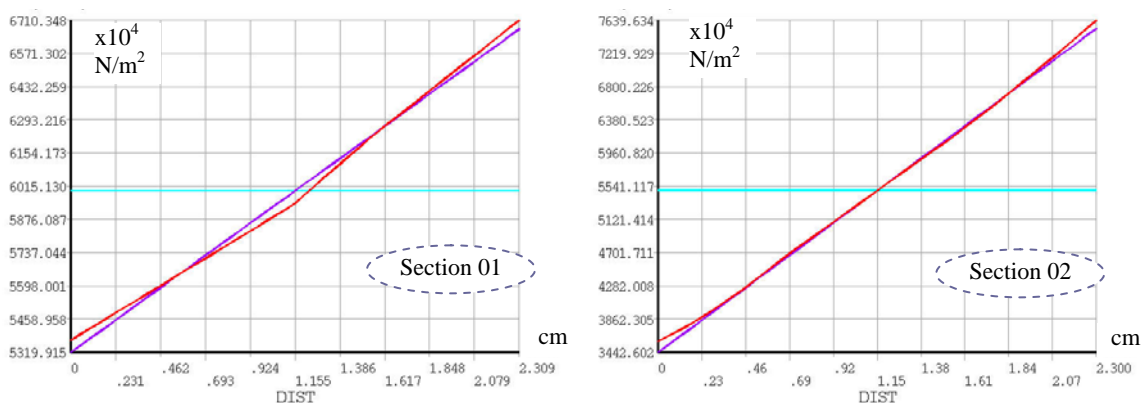


Figure 7a. Stress linearization – Sections 01 & 02

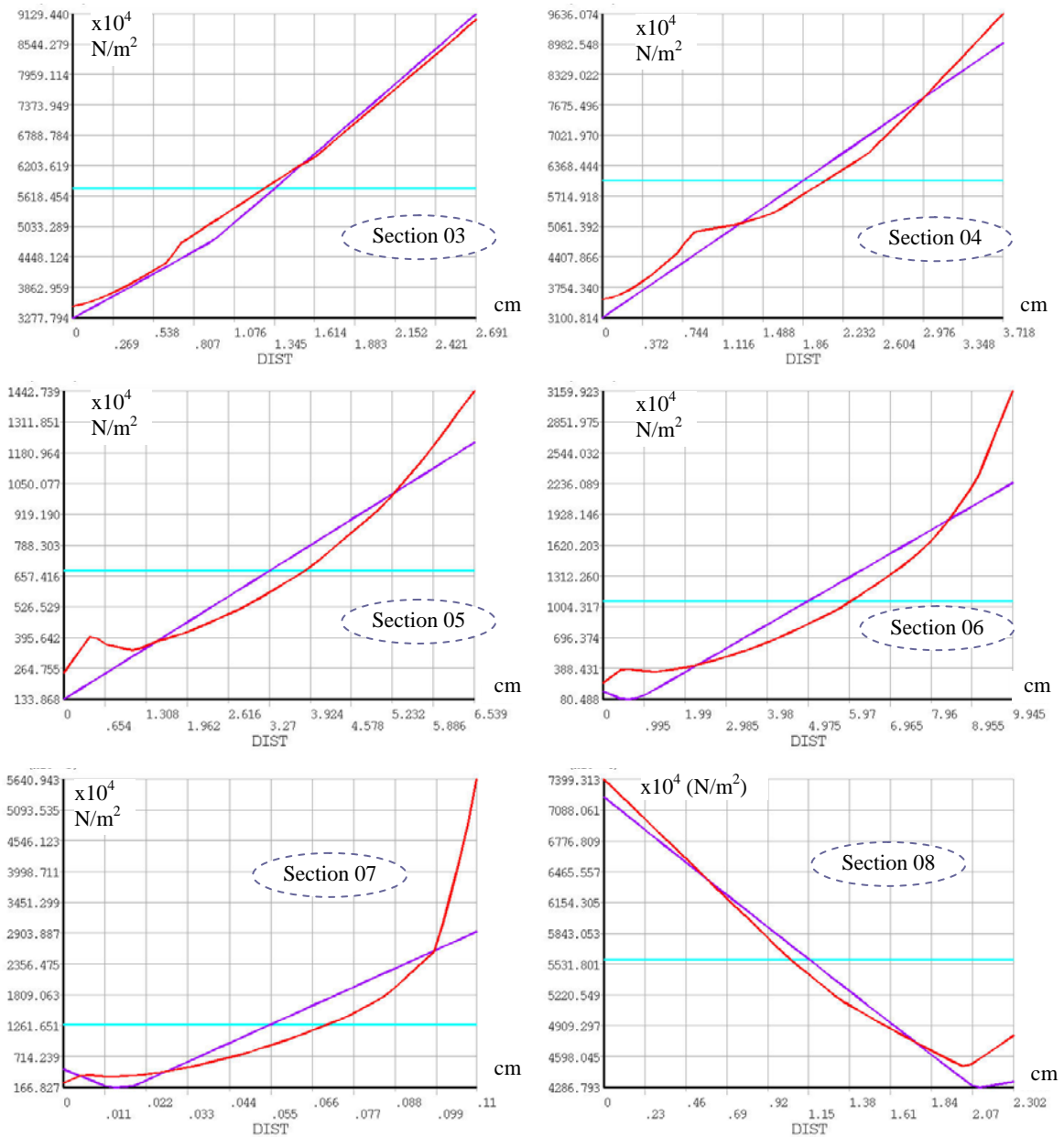


Figure 7b. Stress linearization – Sections 03 to 08

Table 1. Stress Verification (MPa)

Section #	1	2	3	4	5	6	7	8	9	10	Limit
P_m	60.0	---	---	---	---	---	---	---	---	---	110,0
P_L	---	54.8	57.7	60.5	67.9	106.	128.	55.8	48.0	24.2	165,0
$P_L \pm P_B$	67.7	---	---	---	---	---	---	---	---	---	165,0
$\sigma_{actual}/\sigma_{limit}$ (%)	55 41	33	35	37	41	64	78	34	29	15	-----

5. CONCLUSIONS

The paper described the second step of a nuclear piping analysis with a non-standard item when the item should be modeled as a 3D solid with its verification done according to the Sub-section NB 3300 of the ASME Code instead of the NB-3600 one. The used geometry is one that can be found in many plants, from petrochemical to nuclear however, the ‘nuclear focus’ was adopted. Only the primary stresses due to the internal pressure were considered once the scope is to emphasize some of the issues that arise from the stress classification and linearization in discontinuities, which are common in the nuclear area and it is still an open issue. Along with the modeling, analysis and verification a discussion on how to perform the Code verifications was presented, pointing some differences between the present (simplified) analysis, just one load – pressure, and an actual one, with several applied loads.

ACKNOWLEDGMENTS

The authors are grateful to their institution, Nuclear and Energy Research Institute, IPEN-CNEN/SP, for the support given for this work, as well as to Dr. Ana Faloppa for some helpful hints about the English text.

REFERENCES

1. 2007 ASME Boiler & Pressure Vessel Code, Section III, Division 1, Subsection NB – Class 1 Components.
2. 2007 ASME Boiler & Pressure Vessel Code, Section II, Part D – Properties (Materials).
3. Idelchick I. E., *Handbook of Hydraulic Resistance*, 3rd ed., 2005, JPH.
4. ANSYS APDL v11.0, *ANSYS Structural Analysis Guide & Theory Reference Manual*
5. SOLIDWORKS Premium 2010 SP4.0, Dassault Systems.
6. PIPESTRESS *User’s Manual*, Version 3.6.2, DST Computer Services, 2009
7. Albuquerque, L. B., “*Stress Categorization in Nozzle to Pressure Vessel Connections Finite Element Models*” (in Portuguese). MsC Dissertation, 1999, USP/SP.
8. Hollinger, G. L.; Hechmer, J. L., *Phase 1 Report: Three Dimensional Stress Criteria*, PVRC Grants 89-16 and 90-13, New York, NY, 1991.
9. Hechmer, J. L., Hollinger, G. L., *Three Dimensional Stress Evaluation Guidelines Progress Report*, PVP, 1994.
10. WRC 429, J. L. Hechmer, G. L. Hollinger, 1998. “*3D Stress Criteria – Guidelines for Application*”. Welding Research Council, Inc., Bulletin WRC 429, ISSN 0043-2326 / ISBN 1-58145-436-8, Feb. 1998, Welding Research Council, Inc., N. York, USA.