

**Report on Summer Industrial Training at
Heavy Engineering Division (HED) of
Larsen & Toubro Limited, Powai Works**

Prepared by

Subhrajit Bhattacharya

Undergraduate Student, Indian Institute of Technology, Kharagpur
Roll no.: 02ME1041

Under the Guidance of

Mr. N. R. Raykar

Senior Manager, FPEX, HED, L&T Powai Works

Certificate

This is to certify that Subhrajit Bhattacharya, form IIT Kharagpur, has undergone training in the FPEX SBU of HED in L&T Limited., Powai Works for a period of 8 weeks from 5th May to 30th June, 2005, under my guidance. His performance has been satisfactory so as to fulfill all the requirements for successful completion of the training. This document is a bona fide testimony of the trainings undergone and works carried out by him in this course.

Date: _____

Mr. N. R. Raykar,
Senior Manager, FPEX, FP3,
HED, L&T Limited., Powai Works.

1. Introduction

As a part of the curriculum, and for the partial fulfillment of the requirements for completion of the B.Tech degree from IIT Kharagpur, I, Subhrajit Bhattacharya, underwent an industrial training at the HED of L&T Limited., Powai Works for 8 weeks during the two months period of May-June, 2005.

L&T Limited., Powai Works, located at Powai, a place in Mumbai, is a vast industry with wide variety of disciplines. The primary engineering divisions that this industry possess are HED, E&C, ECC and EBG. Among them, HED (Heavy Engineering Division) is a manufacturing unit. In L&T Powai, HED primarily has got three Strategic Business Units (SBU) namely, FPEX, CGPP and RCOG.

FPEX, FP3 was the department in which I was placed for the industrial training. This department is concerned with design and manufacture of Waste Heat Boilers. These equipments have vast application and demand in chemical industries, fertilizer plants, petroleum refineries, etc. The designs of these equipments are performed in multiple stages involving detailed calculations and complex analysis to provide maximum safety. But care also needs to be taken to prevent over designing, which will otherwise increase the material and manufacturing costs as well as the equipment weight making the transportation difficult and expensive.

Hence the HED of Powai Works has a huge department concerned with the designing of the equipments, besides the gigantic workshops where manufacturing of the equipments take place.

Under the guidance of Mr. N.R.Raykar, Senior Manager, HED, I got the opportunity to have exposure to the works carried out in this department of the industry, and hence learn a good deal from them. During the period of two months of industrial training, the works primarily assigned were concerned with finite element analysis and allied topics. A brief discussion on the assignments given and training undergone during this period follows.

2. Assignments completed during training

Below is listed the primary tasks assigned and completed during the period of training. Brief discussions on each of the assignments will follow under the remaining sub-sections of Section 2. The Procedures developed, Documentations made and the Results obtained are described in the Appendices, after Section 2.

1. Study on ASME Section VIII, Division-2, Appendix-4
 - i. Study on stress categorization.
 - ii. Study on definitions of Plastic analysis, Limit analysis, Collapse Load and Shakedown analysis as described in ASME Section VIII, Division-2, Appendix-4 code.
2. Plastic analysis – theory and FE Analysis
 - i. Literature survey on Plastic Analysis.
 - ii. Preparation of procedure for Elastic-Plastic Analysis using Ansys.
 - iii. Preparation of procedure for Shakedown analysis using Ansys.
 - iv. Development of a procedure and a MS Visual Basic code for finding Shakedown Load using Melan's theorem.
 - v. Carrying out Plastic and Shakedown analysis with a pre-solved problem and validation of the results with '*Design by Analysis CD Manual*' issued by European Pressure Equipment Research Council.
3. FE Analysis of a Tube-Tubesheet joint of a Boiler.
 - i. Using Contact elements to simulate thermal and mechanical contact between two surfaces.
 - ii. Performing the solution and analyze the obtain results.
 - iii. Preparing report of the analysis.
4. Working on and developing tools for Material Property Database in MS Access using MS Visual Basic.
 - i. Assisting in the development of a material property database using MS Access based on the ASME Section II, Part D code.
 - ii. Development of a form using VB macro to seek information from the database and interpolating at intermediate temperature points.
 - iii. Development of a form using Visual Basic macro to list, edit and add to the database.
5. Working with universal files of FE softwares I-DEAS and NISA
 - i. Improving the I-DEAS to NISA universal file translator to incorporate Pressure, Convection and Temperature boundary conditions, DOF constraints, and the various element types including first, second and third order axisymmetric, plane, solid and shell elements.
 - ii. Development of a code to convert the MS Access database to I-DEAS universal file.
6. Other short side-assignments
 - i. Theoretical calculation of stresses on the welds and bolts of the support used for holding a pressure vessel in the workshop.
 - ii. Collection of material test data from job files.

2.1. Study on ASME Section VIII, Division-2, Appendix-4 code

Designs in L&T HED are generally done to ensure that they abide by the rules and restrictions imposed by ASME design codes. The ASME codes for design of pressure vessels, heat exchangers and boilers give detailed rules for the design of these components. Stress categorization is one of the most important procedures used for determining whether a stress field produced under a particular load condition violated the allowable limits.

2.1.1. Stress Categorization

In a complex structure under various combinations of loading, it is of great importance to identify the primary, secondary, membrane, bending and peak stresses. These stresses don't have equal effect on the structure and hence their importances are different. Moreover the allowable limits on them are significantly different from each other.

2.1.1.1. Primary, secondary and peak stresses

In order to identify the *primary*, *secondary* and *peak* stresses in a structure, one may proceed according to the definitions given in ASME Section VIII, Division-2, Appendix-4 code. The code makes the major distinction between the primary and secondary stresses by saying that primary stresses are not self-limiting, while the secondary stresses are self-limiting. This implies that the primary stresses are the ones caused by external mechanical loading, while secondary stresses are caused by structural discontinuities, differential thermal expansion, etc. Primary stresses are generally responsible for gross deformation of structures. Secondary stresses are generally self-equilibrating and are of lesser severity than primary stresses.

Peak stresses account for any other causes of stress in the structure, and are generally confined to local regions of the structure. Peak stress may cause fatigue, but not gross deformation of a structure.

2.1.1.2. Membrane and bending

Each of the primary and secondary stresses are again classified as *membrane* and *bending* stresses. To find the membrane and bending stresses, the components of stresses along a particular section of the structure (say, along the thickness of a pressure vessel) need to be *linearized*. That is, if we know a stress component (say σ_{xx}) as a function of l , the distance variable along the path (where l varies from $-L/2$ to $+L/2$, L being the length of the path), we need to express it as,

$\sigma_{xx}(l) = k_1 + k_2 \cdot l + \sigma_{xx,peak}(l)$, where k_1 and k_2 are constants determined using mean and linear regression of $\sigma_{xx}(l)$. Hence, $k_1 = \sigma_{xx,membrane}$ and $k_2 \cdot l = \sigma_{xx,bending}$.

Hence the stress tensors for membrane and bending stresses are obtained along the section. From the stress tensors, applying Tresca's yield criteria, the corresponding stress intensities can be determined. According to ASME Section VIII, Division-2, Appendix-4 code, the following allowable stress limits have been imposed on the stress intensities,

Primary membrane (P_L) + Primary bending (P_b) stresses — $1.5kS_m$.

Primary ($P_L + P_b$) + Secondary (Q) — S_{PS}

Primary ($P_L + P_b$) + Secondary (Q) + Peak (F) — S_a

Where, i) S_m is the allowable stress for the material as given in ASME Section II, Part D, Table 2A and 2B; ii) k is a factor depending on the types of loading (e.g. internal pressure, wind load, earthquake, etc.) and is given by Table AD-150.1 of Section VIII, Division-2, Appendix-4; iii) S_{PS} is the maximum of $3S_m$ and $2S_y$, where S_m and S_y are the tabulated allowable and yield stresses respectively, averaged on the highest and lowest operating temperatures; iv) S_a is obtained from the fatigue curves given in Section VIII, Division-2, Appendix-5 and depends on the ultimate tensile strength of the material and the number of operational cycles for the equipment.

2.1.2. Plastic Analysis

According to ASME VIII Division 2 Appendix 4-136.5, *“The limit of general membrane stress intensity... local membrane stress intensity... and primary membrane plus primary bending stress intensity... need not be satisfied at a specific location if it can be shown that the specified loadings do not exceed two-thirds of the plastic analysis collapse load determined by application of 6-153...”*

However, the procedure for determination of collapse load according to Appendix 6-153 is based on some experimental analysis of the actual structure, which hence can't be used for theoretical predictions or Finite Element Analysis. Though any further clarification of the procedures for Plastic Analysis has not been provided in the code, the above-mentioned rule shows prospective potential for performing plastic analysis of a structure.

An alternative to Plastic Analysis is Limit analysis, which has been described under ASME VIII, Division 2, Appendix 4-112 as *“a special case of plastic analysis in which material is assumed to be ideally plastic”*. In Appendix 4-136.3, Limit Analysis is described as, *“The limit on general membrane stress intensity... local membrane stress intensity... and primary membrane plus primary bending stress intensity... need not be satisfied at a specific location if it can be shown by limit analysis that the specified loadings do not exceed two-thirds of the lower bound collapse load.”* This ‘Collapse Load’ for limit analysis is basically the ‘Lower bound Collapse Load’, which is described in 4-112 as, *“...the maximum load a structure made of ideally plastic material can carry. The deformations of an ideally plastic structure increase without bound at this load, which is termed collapse load.”*

As mentioned before, no specific procedure for Plastic analysis was available from the code. Hence, further investigation into the subject was encouraging. The result of the literature surveys and understandings on Plastic Analysis has been described next in Section 2.2.1.

2.2. Plastic analysis – theory and FE Analysis

The ASME, Section-VIII, Division-2, Appendix-4 code gives detailed procedure for calculation of stress intensities and stress categorization. However the permissible limits set to the different categories of stresses are completely bases on elastic analysis and ensures that using such limits won't let the material cross the yield point. However this often leads to over-designing. Logically, some parts of the vessel may be allowed to go beyond the yield point of the material as long as it remains bounded in volume by surrounding materials in elastic region. Though this would cause some portions of the structure to get into plastic region, but will still not allow failure or gross deformation of the structure. Plastic analysis aims at such designing allowing plastic behavior of some parts of the structure.

As mentioned in the previous Section, the ASME code requires the determination of 'collapse load' for the plastic analysis. Hence it was also of great interest to find a method for determining collapse load of a structure.

2.2.1. Literature survey on Plastic Analysis

A study was performed on Plastic Analysis and informations on already existing theories and recent developments on the field were gathered. The primary sources of the informations were some textbooks on mechanics if materials and *ASME Journal of Pressure Vessel Technology*. This helped in the further studies and analysis. A short account on the results of the studies follows.

2.2.1.1. Lower Bound Collapse Load

ASME code describes this load as “...the maximum load a structure made of ideally plastic material can carry. The deformations of an ideally plastic structure increase without bound at this load, which is termed collapse load.” This load is basically such that it produces a limiting statically admissible system of stresses that satisfies equilibrium conditions and stress boundary conditions. The actual collapse load is always greater than the Lower Bound Collapse Load.

2.2.1.2. Upper Bound Collapse Load – a theoretical determination technique

In a system where the loading is dynamic and cyclical, the Upper Bound Collapse Load is defined as the Kinematically admissible system, obtained by selecting the displacement/strain rate system (e.g. the St. Venant's equation), and determining the flow rule. This gives,

$$\text{kinematically admissible load} = \frac{\text{internal work}}{\text{unit external work}}$$

Where, the 'internal work' is the internal work done under some extreme load condition, and the 'unit external work' is the work of unit load on a selected displacement rate system. The method for determining this load is illustrated using a simple example of a thin shell. Let's consider a small element of the shell of thickness t and the material Modulus of Elasticity Y . As shown in the figure, the element is subjected to membrane stresses N_θ and N_ϕ , and bending moments M_θ and M_ϕ .

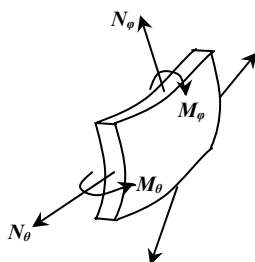


fig - 1

For finding nondimensional stresses, we define $N_0 = Yt$ and $M_0 = Yt^2/4$, and $n_\theta = N_\theta / N_0$, $n_\phi = N_\phi / N_0$, $m_\theta = M_\theta / M_0$, $m_\phi = M_\phi / M_0$.

Then the stress vector is defined by, $\mathbf{S} = (n_\theta, n_\phi, m_\theta, m_\phi)$.

Next to find the strain rate vector, we first need to assume a displacement rate field. The displacement rate field may be calculated from the information of amplitude and frequency of loading/unloading. Let this displacement rate vector be $\dot{\mathbf{U}}$. From this, the strain rate vector may be calculated the same way the strains are calculated from displacements.

Let this strain rate vector be $\dot{\mathbf{E}} = (\dot{\varepsilon}_{\theta n}, \dot{\varepsilon}_{\phi n}, \dot{k}_\theta, \dot{k}_\phi)$, where, $\dot{k}_\theta = \dot{\chi}_\theta \frac{t}{4}$, $\dot{k}_\phi = \dot{\chi}_\phi \frac{t}{4}$;

$\dot{\chi}_\theta$ and $\dot{\chi}_\phi$ being the angular velocities. Then the internal work will be given by,

$$\dot{W}_{int} = \int_{\text{whole structure}} N_0 \mathbf{S} \dot{\mathbf{E}} dV$$

and the unit external work is given by,

$$\dot{W}_{ext} = \int_{\text{whole structure}} \dot{\mathbf{U}} dV.$$

Hence, kinematically admissible load = $P_{ka} = \dot{W}_{int} / \dot{W}_{ext}$.

Proceeding in the above mentioned manner, expressions for collapse loads have been derived for various structures in the book 'Pressure Vessel Design & Analysis' by Bickel and Ruiz.

2.2.1.3. A theoretical technique for calculation of Plastic Collapse Load

This technique, as described in the book by Bickel and Ruiz, is based on the fact that at the collapse load, small increment in the external load will cause infinitely large deformations in the structure, hence cause infinitely large changes in strain. Hence, the primary aim of this technique is to find a relation between the external load (P) and a characteristic strain for the structure (e.g. may be strain at the internal or external circumference of a pressure vessel). Below is given a schematic description of how this method can be applied to find collapse load for a cylindrical pressure vessel.

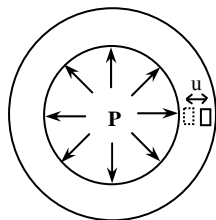


fig - 2

If u = radial displacement at a point; and ε_r and ε_θ are the logarithmic radial and hoop stresses, then,

$$\frac{\partial u}{\partial r} = e^{\varepsilon_r} - 1 \quad \text{and} \quad \frac{u}{r} = e^{\varepsilon_\theta} - 1.$$

For incompressibility, $\varepsilon_r = -\varepsilon_\theta$.

Two quantities called significant stress and significant strain are defined as,

$$\bar{\sigma} = \sigma_\theta - \sigma_r, \quad \text{and}$$

$$\bar{\varepsilon} = -\frac{2}{\sqrt{3}} \varepsilon_r = \frac{2}{\sqrt{3}} \varepsilon_\theta,$$

which are related to each other by the stress-strain curve. [e.g. in case of perfectly linear and elastic material $\bar{\sigma} = E \cdot \bar{\varepsilon}$.]

From the above equations, eliminating u ; expressing σ_θ in terms of σ_r and $\bar{\sigma}$; and expressing ε_r and ε_θ in terms of $\bar{\varepsilon}$, the equation obtained is given by,

$$d\sigma_r = \frac{\bar{\sigma} \cdot d\bar{\varepsilon}}{1 - e^{\sqrt{3}\bar{\varepsilon}}}$$

Now, using boundary conditions $\sigma_r = -P$, $\bar{\varepsilon} = \varepsilon_i$ at $r = r_i$; and $\sigma_r = 0$, $\bar{\varepsilon} = \varepsilon_o$ at $r = r_o$, for this specific example, we obtain,

$$P = \int_{\varepsilon_i}^{\varepsilon_o} \frac{\bar{\sigma} \cdot d\bar{\varepsilon}}{1 - e^{\sqrt{3}\bar{\varepsilon}}}$$

Considering that collapse occurs when the strain at inner circumference increases indefinitely, the collapse point is given by,

$$\frac{\partial P}{\partial \varepsilon_i} = 0.$$

Hence, knowing the stress-strain relation curve, it is possible to determine exactly the load (P) at which collapse occurs.

2.2.1.4. First Yield Load and Limit Load for a thick-walled pressure vessel

As described in ‘Pressure Vessel Design Concepts and Principles’ by J. Spence and A.S.Tooth, a pressure vessel under internal pressure assuming elastic-perfectly plastic material and Tresca’s Yield criteria, some simple calculations involving equilibrium considerations give,

The load at which the material at the inner circumference first crosses the Yield point

$$= \text{First yield load} = P_y = \frac{\sigma_y}{2} \left[1 - \left(\frac{r_i}{r_o} \right)^2 \right].$$

The load at which the whole material of the vessel cross yield point and the vessel

$$\text{collapses by expanding indefinitely} = \text{Limit load} = P_L = \sigma_y \cdot \ln \left(\frac{r_o}{r_i} \right).$$

These expressions are useful for quick and approximate calculations.

2.2.1.5. Limit Load Multipliers and their significances

In the paper entitled “*A Finite Element Technique for Limit Analysis of Structures*”, *J. Pressure Vessel Technology, Vol-112, p-138*, quantities (denoted by m) called limit load multipliers have been defined. Then the Lower bound theorem states $m_s \leq m^*$, and the Upper bound theorem states $m_k \geq m^*$, where the limit load multipliers are defined as,

$$m_s = \sqrt{\frac{k^2}{f_{\max}}}, \text{ where, } f_{\max} \text{ is the maximum value of the quantity } f = \frac{1}{2} \sigma^T \underline{c} \sigma \text{ within}$$

the structure, with σ as the stress components expressed in a vector form, and \underline{c} being a constant matrix, k being a constant.

And, $m_k = \frac{1}{V} \int_V g^{1/2} dV$, the integration being done over the entire volume of the structure, where, $g = 2\varepsilon^T \underline{c} \varepsilon$, with ε as the strain components expressed in a vector form.

And m^* is the multiplier which actually gives the collapse load.

From the above, the following conclusions can be drawn,

- The Lower bound limit load is dependent on the stress conditions in the structure, and is inversely proportional to the square-root of the quantity f_{max} , which is a measure of the maximum total stress in the structure.
- The Upper bound limit load is dependent on the strain conditions in the structure, and is proportion to the mean value of g , which is a measure of total strain.
- If the values of m_s and m_k are plotted with the load, the point of intersection of the two plots will indicate the collapse load.

2.2.1.6. Shakedown or cessation of plastic deformation (work-hardening)

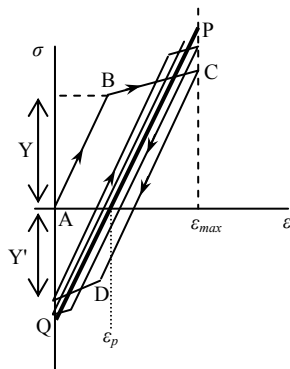


fig - 3

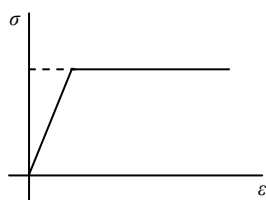


fig-4

As shown in the adjacent stress-strain plot (*fig-3*), due to repeated loading and unloading between $\varepsilon = 0$ and $\varepsilon = \varepsilon_{max}$, the material follows the path ABCD... and finally approaches the straight line PQ with a permanent plastic strain of ε_p and the maximum strain of ε_{max} . This phenomena is known as shakedown. Generally, it is assumed, $\varepsilon_{max} \leq 2Y / E$. Yield strength at compression (Y') \leq Yield strength at tension (Y).

A very special case of the stress-strain curve described above is that of elastic-perfectly plastic material. The corresponding stress-strain relation is plotted in *fig-4*. Though this is a highly idealized and approximate representation of the actual material behavior, an elastic-perfectly plastic material model performs fairly well when it comes to elastic-plastic and shakedown analysis of a structure.

2.2.1.7. A rule for ratcheting analysis of structures under cyclic variation of secondary stresses using concept of Primary Effective Stress

This method, as described in the paper entitled "*A Design Rule for Thermal Ratcheting*", *J.Pressure Vessel Technology, Vol-108, p-188*, involves the evaluation of a quantity called *Primary Effective Stresses* (P_{eff}) and comparison of this quantity with the maximum stress in the structure. Below is described the steps involved in evaluating P_{eff} at a particular point in the structure:

- i. Let the structure be undergoing cyclic external loadings. Elastic analysis of the structure is performed under the different extreme load conditions (say,

fully loaded and no load conditions). Hence the stress fields are obtained at these loadings.

- ii. Distinguish between the primary and secondary stresses at each of these loading conditions.
- iii. Now, considering a particular point in the structure, let the maximum intensity (i.e. Tresca's Equivalent) of primary stress at the point be P .
- iv. Let the range of secondary stress intensity variation be QR .
- v. The *Secundarity Ratio* is then calculated as, $SQ = QR / P$.
- vi. Once the value of SQ is obtained, one need to consult the *Efficiency Diagram* (a plot between SQ and V , determined on experimental basis, and is given in *J.Pressurs Vessel Technology, fig-15, Vol-108, p-194.*) to determine the *Efficiency Index*, V .
- vii. The Effective Primary Stress is hence calculated as, $P_{eff} = P / V$.

This P_{eff} is a fictitious quantity and is basically the stress, which on application for the same time gives strains as that of the real cyclic loading.

The condition for no ratcheting to take place is generally taken as $P_{eff} \leq 1$ to $1.2 S_m$.

2.2.1.8. Detailed Elastic-Plastic Analysis of thick-walled cylindrical vessels

In the group of papers entitled "*Elastic-Plastic Analysis of Tubes*", *J.Pressure Vessel Technology, Vol-114, p-213*, the author W.Jiang has performed detailed theoretical calculations for thick cylinders under various types of cyclic loadings and has deduced various modes of shakedown. In this paper, the term 'shakedown' has been used in a more generalized sense and has referred to any steady-state response of a system under any cyclic loadings. The primary features of the analysis performed in the paper are,

- Detailed expressions for σ_θ , σ_r and τ have been derived for cylinders with internal and external pressures and also rotating with a constant angular velocity.
- The analysis apparently uses von Mises's yield criteria.
- For shakedown analysis, two loads are assumed (upper and lower) and the stresses are determined at each of them.
- 9 different modes of shakedown (i.e. 9 solutions for the differential equations depending on the boundary conditions, loads and the geometric parameters) has been deduced – 1 purely elastic, 2 elastic, 3 reversed plastic and 3 ratcheting.
- Ratcheting analysis has also been performed, where the values of σ_θ , σ_r and τ have been derived at the k^{th} cycle of loading/unloading. It was then checked whether those values remain bounded as $k \rightarrow \infty$.

2.2.1.9. An alternative method for finding Lower bound Limit Load (*J.PVT, vol-116, p-340*)

Let a structure be subjected to a cyclic loading. Let a load P_d be applied on the structure at a particular instant of time. The corresponding stress field be, σ_s , determined from analysis of the structure. The Lower bound Limit Load is then given by,

$$P_l = \max_t \left(P_d \frac{\sigma_y}{|\sigma_s|_{\max}} \right)$$

where, the function $\max_t ()$ denotes maximum of all the values inside the brackets obtain at various instants of time, i.e. maximum over time; While $|\sigma_s|_{\max}$ is the maximum stress intensity inside the structure at a particular time instant, i.e. maximum over space; And σ_y is the Yield strength.

2.2.1.10. Melan's Theorem – Determination of Shakedown Load from Limit Collapse Load

The Melan's theorem is base on determination of a self-equilibrating stress field, $(\sigma_{ij})_{res}$ defined as,

$$(\sigma_{ij})_{res} = (\sigma_{ij})_{ep,l} - (\sigma_{ij})_{le,l}$$

Where, $(\sigma_{ij})_{ep,l}$ is the stress field for the structure made up of elastic-perfectly plastic material under the Limit load or the Lower bound Collapse Load (l), and $(\sigma_{ij})_{le,l}$ is the stress field under the same load (l) but using a linearly elastic material model. Once the residual stress field is determined, it is multiplied by a factor ' β ' to bound the stress field,

$$(\sigma_{ij})_{res,co} = \beta \cdot (\sigma_{ij})_{res}$$

such that $(\sigma_{ij})_{res,co}$ does not violate the Yield criteria at any point in the structure. β may be 1 when the uncorrected self-equilibrating stress field does not violate yield conditions. Then the shakedown load is the limit collapse load.

After this, the corrected stress field $(\sigma_{ij})_{res,co}$ is superimposed on a scaled version of the linearly-elastic stress field such that it does not violate the yield condition,

$$(\sigma_{ij})_{SD} = (\sigma_{ij})_{res,co} + \alpha \cdot (\sigma_{ij})_{le,l}$$

where, α is the scaling factor. The value of α to be taken should be the highest possible one for which $(\sigma_{ij})_{SD}$ does not violate the yield conditions.

Then, if P_{lim} denotes the limit collapse load, then the Shake-down load is determined by,

$$P_{SD} = \alpha \cdot P_{lim}$$

The application of Melan's Theorem has been described later in procedure for determination of shakedown load.

2.2.2. Preparation of procedure for Elastic-Plastic Analysis using Ansys – to determine Limit Collapse Load

The Finite Element Analysis software Ansys 8.0 was extensively explored to determine how elastic-perfectly plastic material properties are declared and the non-linear analysis is performed. The main objective was to determine the 'Lower bound collapse load' – the load at which gross deformation of the structure is about to begin. By using an elastic-ideally plastic material model, this particular load was determined by the load at which numerical instabilities begin and the solution starts diverging. The detailed stepwise procedure developed as a part of the given assignments during the training has been described in Appendix-A.

This procedure has then be used to find the collapse load for a problem described in the 'Design by Analysis CD Manual' issued by European Pressure Equipment Research Council, University of Strathclyde. In that CD Manual, the collapse load for the structure has already been determined by some alternative route. The results

obtained by the method described in Appendix-A were in satisfactory agreement with those of the manual. The details of the results obtained can be found in Appendix-F.

2.2.3. *Preparation of procedure for Shakedown analysis using Ansys – to determine whether shakedown takes place at a particular load condition*

To determine whether Shakedown takes place under a given load cycle, Ansys 8.0 was used to perform the solution (static or transient) at several load-steps to simulate the cyclic loading and unloading. The details of the procedure developed is given in Appendix-B.

This procedure was used to check whether Shakedown takes place at the Shakedown load determined by Melan's Theorem (section 2.2.4.). Shakedown was found to take place at that load. Appendix-F gives the details of the results obtained.

2.2.4. *Development of a procedure and a VB code for finding Shakedown Load using Melan's theorem*

As described in section 2.2.1.10., the Melan's Theorem requires a fairly large amount of calculations involving the stress fields. Initially it was attempted to perform the calculations using Ansys Element Tables. But on finding that this leads to lots of time consuming complications, some alternative way was searched for.

The easiest alternative solution found was to export the stress fields in ASCII file format from Ansys and directly operate with them using a computational code written in some programming language to find the values of α and β .

Hence, a code was written in MS Visual Basic which would take input the exported stress fields $(\sigma_{ij})_{ep,l}$ and $(\sigma_{ij})_{le,l}$ from Ansys and the Yield strength value, and will output the values of α and β .

This program was used with the same problem of 'Design by Analysis CD Manual'. The values of α and β have also been evaluated in the CD manual using some other methods. The values obtained using the VB program was found to be in excellent agreement with those of the CD Manual.

The attempted procedure for Melan's theorem evaluation using Ansys Elements Table is given in Appendix-C. The procedure for exporting the stress fields from Ansys and using the VB code to determine α and β is given in Appendix-D. The pseudo-code used in the VB program has been described in Appendix-E. The verification of the values of α and β obtained from the code is given in Appendix-F.

2.2.5. *Carrying out Plastic and Shakedown analysis with a pre-solved problem*

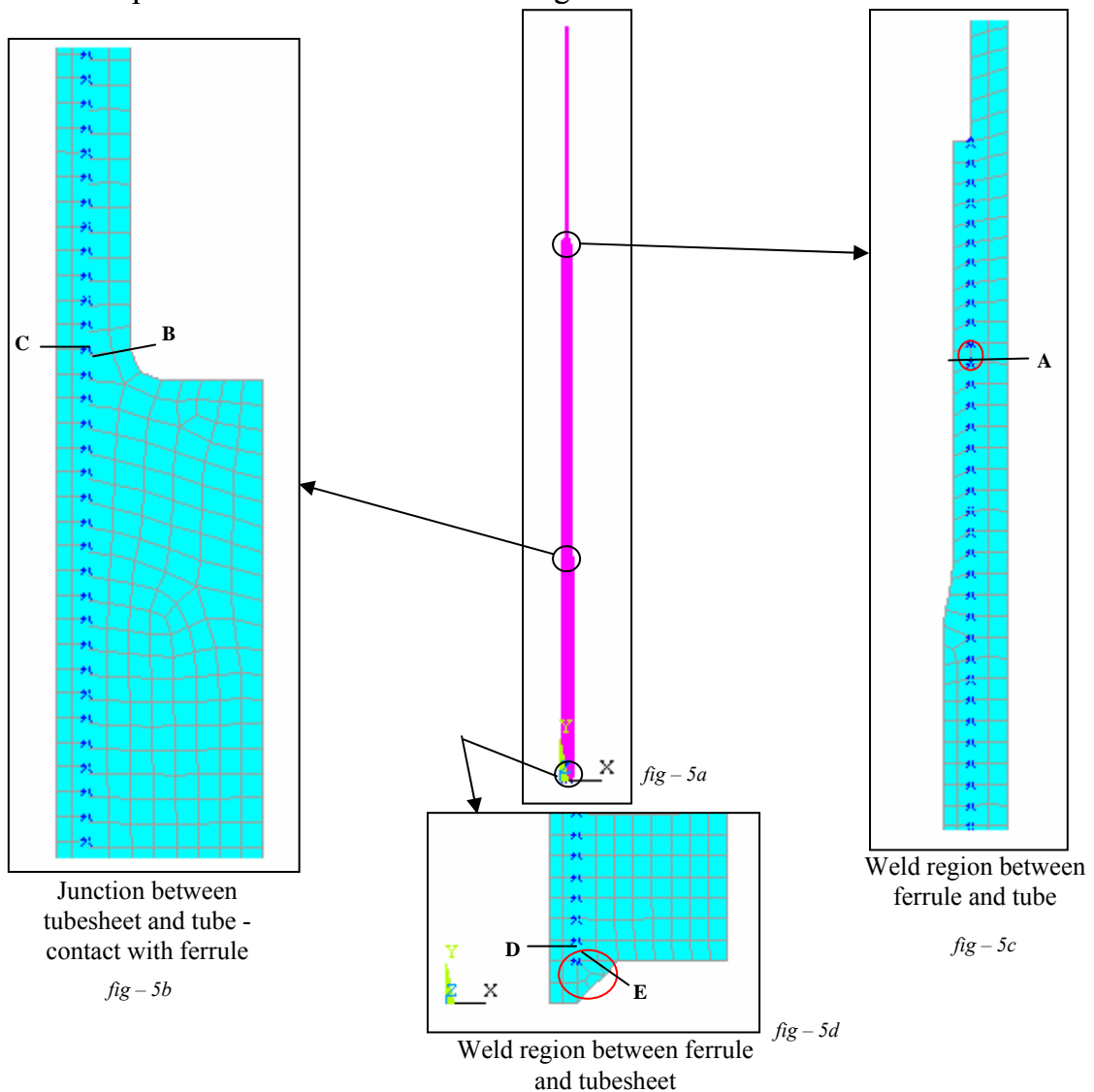
As mentioned earlier, the procedure for determining Limit Collapse Load (Appendix-A), Shakedown Load using Melan's Theorem (Appendix-D), and verifying whether Shakedown takes place at that load (Appendix-B) has been validated by re-solving a FE problem already solved in 'Design by Analysis CD Manual' issued by European Pressure Equipment Research Council, University of Strathclyde. The results obtained were in good agreement with those of the CD Manual. The details are described in Appendix-F.

2.3. FE Analysis of a Tube-Tubesheet joint of a waste heat boiler

The boiler studied was basically a shell and tube type heat exchangers which consist of several U-shaped tubes attached to a tubesheet. Through these tubes hot corrosive gasses flow, and comparatively cooler water that flow over the tubes. The whole assembly resides inside a pressure vessel. The analysis aimed at simulating an axisymmetric model of a single tube attached to the tubesheet and investigation of whether the allowable stress intensity limits are exceeded at any portion of the model.

2.3.1 Using Contact elements to simulate thermal and mechanical contact between two surfaces

There is a layer of Inconel 600 material at the inner surface of the tube, which has been shrink-fitted during the manufacture process. This inside lining is called the Ferrule and is joined to the tubesheet at the bottom by a fillet weld, and is joined to the tube near the top by a small weld region. The remaining of the ferrule is not attached to the tube or tubesheet, hence there can be relative sliding motions between the outer surface of the ferrule and inner surface of the tube-tubesheet. In order to simulate this situation, contact elements had to be used to establish thermal contact and sliding mechanical contact between the two surfaces. The figures below give the schematic representation of the FE model along with the contact elements.



The dark blue spots between the ferrule and tube-tubesheet show the contact elements. The red circles marks the weld regions. The lettered sections are the places where stress linearization has been done.

2.3.2. *Solving the model and obtaining the results*

The model was created according to the dimensions given in the drawings for the Boiler. The materials and material properties chosen were according to the drawings itself. For the purpose of obtaining the material properties, ASME Section II, Part-D codes were used. The contact elements were generated between the appropriate surfaces.

Appropriate boundary conditions were applied and the solution was performed. Because of presence of contact elements, the solutions were non-linear in nature.

The details of the Elements used, Material Properties and Boundary conditions applied are given in Appendix-G.

2.3.3. *Interpreting the results and drawing conclusions*

The results obtained are described in Appendix-H. The stresses were linearized and categorized into membrane and bending stresses at some critical sections. From the obtained results the following conclusions could be made:

- The categorized stresses are within the allowable limits in accordance to ASME Section VIII, Division-2, Appendix-4 code.
- The maximum stress intensities are obtained near the weld regions.
- The contact elements showed desired behaviors. They established thermal contact by showing continuity in temperature across the interface. Moreover Normal reaction was also transmitted between the surfaces which was reflected by the continuity in the x -component of displacement and the σ_{xx} component of the stress across the interface. However, sliding was allowed between the surfaces, which were reflected by the discontinuities in y and z component displacement plots.
- The welds were modeled successfully as all the quantities showed continuity across the welds.

2.4. *Material Property Database in MS Access*

ASME Section II, Part-D code is used extensively to obtain the property values for material for both *Design by Formula* and *Design by Analysis*. But there are some inherent disadvantages with reading from the code:

- Owing to the massiveness of the code with thousands of materials and their property tables, reading property values from the code is not only difficult and time-consuming, but also is highly prone to errors.
- The units for material properties used in the code are all in FPS system and the temperatures are in Fahrenheit. Hence, every time the properties are read from the code, they need to be converted to the desired unit systems. This is rather cumbersome and may give rise to errors.
- The code provides the property values at some particular temperature points. In order to obtain the property values at some intermediate temperature, interpolation is required. Performing the interpolation manually every time is again tedious and prone to errors.

Hence it would of great help if the material properties can be inputted into a computer database and then read, interpolated or even modified in any convenient unit system using a suitable computer program. This was exactly the target of this project and has been finally achieved successfully.

2.4.1. *Assisting in development of the material property database using MS Access*

The first step towards the development of the database was to design a structure for the database. After a considerable amount of discussions, thoughts and experimentations, a final structure for the database in MS Access was decided. The description of the structure of the database follows.

The database makes use of one main table, 'Materials', containing the material names, compositions and references to parts of the 'Data_' tables containing the property values. The 'Data_' tables may be viewed as pools (or collections) of several sets of property values, to which references are made in the 'Materials' table.

There is also a set of 'ID_' tables, whose soul purpose is to create one-to-many relationships between the 'Materials' and 'Data_' tables, which would otherwise had been many-to-many relationships. The user of the database may not be concerned much about the 'ID_' tables.

Below are the salient features of the structure of the database.

- The Field 0 ('Material Name') of the 'Materials' table contains the names of the different materials, and Field 6 ('Composition') contains the composition.
- For each record in the 'Materials' table, the Fields 1, 2, 3, 4 and 5 contains a reference number or 'ID' of the properties 'Modulus of Elasticity', 'Thermal Conductivity', 'Coefficient of Thermal Expansion', 'Thermal Diffusivity' and 'Allowable Stress' respectively. The ID refers to a particular set of values in the respective 'Data_' table which are the property values of the material.
- There are 5 'Data_' tables which contain several sets of the property values. Each set (identified by an ID) contains the property values at different temperature points.
- All the units used in the database are same as those in ASME code in Section II Part D. Temperatures are in Fahrenheit.

A particular data set may be referenced by more than one material. Making changes in that data set will hence cause change in all those materials' that property.

Notes:

- i) A 'Record' refers to an entry in a table.
- ii) The term 'Field n' refers to the column (n+1) of the particular table or record.
- iii) The term 'property data set' refers to a set of property values at different points of temperature. Each material property 'Data_' table contains several such sets, which in turn are referenced by the materials in the 'Materials' table to describe the corresponding property. For a particular property, a particular set has an unique ID number.

The following diagram makes the structure of the database clear:

Material Name	Modulus of Elasticity	Thermal Conductivity	Thermal Expansion Coefic	Thermal Diffusivity	Allowable Stress
SA 182 GR F1	2	3	1	3	2
SA 182 GR F11	4	3	1	3	3
SA 182 GR F22	5	4	1	4	4
SA 204 GR B	2	3	1	3	5
SA 213 T11	4	3	1	3	6
SA 213 T12	4	3	1	3	7
SA 213 T22	5	4	1	4	8
SA 302 GR B	2	3			9
SA 302 GR D	2	3			10
SA 387 GR 11 CL2	4	3			11
SA 387 GR 12 CL2	4	3			12
SA 387 GR 22 CL3	4	3			13
SA 508 GR 3 CL1	3	3			14
SA 516 CD 70	1	1			15

ID	Temperature	Modulus of Elasticity
3	1400	17.7
4	70	29.6
4	200	29
4	300	28.5
4	400	28
4	500	27.4
4	600	26.9
4	700	26.2
4	800	25.6
4	900	24.8
4	1000	23.9
4	1100	23
4	1200	21.8
4	1300	20.5
4	1400	18.9
5	70	30.6
5	200	29.9

fig – 6: Structure of the database – Displaying a property data set – Modulus of Elasticity values for material SA 182 Gr F11

The procedure documented for working with the database manually has been described in Appendix-I.

2.4.2. Developing VB macros to work with the database

The task of creating VB macros to read, interpolate and modify the database was next assigned. It was a bit difficult task to interact with MS Access using the Visual Basic macros because no proper help providing the desired informations was available in the help files. However, taking help of VB Object Browser all the tasks were finally completed.

The descriptions of the two forms developed in the process along with their screenshots follow:

2.4.2.1. The 'Seek/Interpolate' form

Material Name: SA 182 GR F22
Material Property: Modulus of Elasticity Unit: kgf/(mm²)
Temperature: 300 Unit: °C Seek

Material Property value at a table temperature just greater or equal to than the given temperature = 1.9476E+04 at temperature 315.556
Material Property value at a table temperature just lower than or equal to the given temperature = 1.9898E+04 at temperature 260.000
Interpolated Material Property value at the given temperature = 1.9594E+04

Open 'Materials' Table Open Property Value database Table

Record: 1 of 1

fig - 7

The salient features of the form are:

- The user may select the required Material Name and Material Property from the drop-down lists.
- The user may chose to view the output values of the material property in any desired unit system available in the drop-down list. The units of temperature can be chosen between C, F or K.
- The user may click the 'Open Materials Table' button to open the 'Materials' table from the actual database at the position of the selected material.
- The user may click the 'Open Property Value database Table' button to open the corresponding 'Data_' table from the actual database at the position where the presently displayed property values reside.

The detailed procedure for working with this form was documented, and is given in Appendix-J.

2.4.2.2. The 'List/Edit/Add' form

The screenshot shows a software window titled "List/Edit/Add : Form". At the top, there are input fields for "Material Name" (SA 213 T12), "Composition" (1Cr - (1/2)Mo), "Material Property" (Thermal Conductivity), "Material Property unit" ((kgf-mm)/(s-m)), and "Temperature unit" (°F). An "Add Material" button is to the right. Below these is a table titled "List (loadad property set):" with columns "Temperature" and "Property Value". The table contains 11 rows of data. To the right of the table are buttons for "[<--] Update Record", "[<--] Add Record", and "[X] Delete Record". Further right are input fields for "Temperature" (70 .000) and "Property Value" (4 .18106), with a "Units: As selected for the listing." label. Below the table are "Sort List" and "Clear List" buttons. At the bottom, there is a "Save this property data set with ID number:" field (3) and an "Update Database with the current list" button. The status bar at the very bottom shows "Record: 1 of 1".

Temperature	Property Value
70 .000	4 .18106
100 .000	4 .16342
150 .000	4 .14578
200 .000	4 .14578
250 .000	4 .12813
300 .000	4 .12813
350 .000	4 .11049
400 .000	4 .07521
450 .000	4 .05757
500 .000	4 .00464
550 .000	3 .96936
600 .000	3 .91644
650 .000	3 .86351
700 .000	3 .81059
750 .000	3 .75766

fig - 8

The salient features of the form are:

- The user may select the required Material Name and Material Property from the drop-down lists.
- The user may chose the desired units for the material property and temperature to work with.
- The user may modify or delete existing records in a property value data set, and also add new record to the database.
- The user may add new materials to the database by typing the new name in the 'Material Name' box, which will activate the 'Add Material' button.
- The user may select from the existing compositions for the new material, which will accordingly assign IDs to some of the properties, or he/she may chose to enter a new composition.
- The user may assign IDs from existing data sets to a property of a new material.

The documentation on the detailed procedure for working with this form is given in Appendix-J.

2.5. *Working with universal files of I-DEAS and NISA*

Beside Ansys, I-DEAS and NISA are two Finite Element softwares which are used extensively for FE Analysis in the industry. This part of the project involved working with the universal files of these two softwares in order to interact with them and among them.

The Universal Files are basically ASCII data files containing informations about the elements, nodes, constraints, boundary conditions, element types and material properties used in a FE model, placed under several blocks (bound by delimiters) or headers. Hence they can be read or modified on having a knowledge about the different dataset identifiers or headers and the formats in which the data are written in the Universal Files. These informations are documented in the I-DEAS Files Format Reference Guide manuals for I-DEAS universal files, and in the NISA users' manual for the NISA universal files.

2.5.1. *Improving the I-DEAS to NISA universal file translator*

I-DEAS and NISA have got their own pros and cons. As for example, I-DEAS is excellent when it comes to modeling and meshing of complex structures because of its user-friendly and interactive modeling environment. Whereas the modeling environment of NISA is extremely poor and it takes lot of efforts even to view the model properly. On the other hand, with the present license of the I-DEAS that is being used, it is not possible to perform nonlinear analysis. But NISA can perform nonlinear analysis very well.

Hence it was desired to find a way so that the modeling could be done in I-DEAS, while the solution can be performed in NISA. The possible way of making them interact with each other was their universal file formats for their models. But the Universal file formats of I-DEAS and NISA are remarkably different. Hence it was required to create a translator to convert the I-DEAS universal file to NISA universal file. Such a translator was already created and was in its elementary stage that could convert the informations about the nodes and elements. The task assigned regarding this translator was to improve it so that it can translate the informations about boundary conditions, loads, DOF constraints and different element types. This was achieved and the translator was improved so that it can also translate the following informations:

- The type of analysis – Static or Thermal
- Edge and Face Convection boundary conditions [I-DEAS Unv dataset no. 790; NISA dataset '*CONVBC'].
- Face Heat Flux boundary conditions [I-DEAS Unv dataset no. 790; NISA dataset '*DEFLUX'].
- Face Pressure [I-DEAS Unv dataset no. 790; NISA dataset '*PRESSURE'].
- Displacement restraint [I-DEAS Unv dataset no. 791; NISA dataset '*SPDISP'].
- Temperature boundary conditions [I-DEAS Unv dataset no. 792; NISA dataset '*NDTEMPER'].
- Element types (includes plane, axisymmetric, solid and shell elements of first, second and third orders) [NISA dataset '*ELTYPE'].

2.5.2. *Developing code for converting the MS Access materials' database to I-DEAS universal file*

While performing FE analysis using I-DEAS, it is often required to input the material properties at different points of temperature so as to consider the variation of the material properties with temperature during the analysis. As the database created in MS Access contained the material properties at various points of temperature, it was of interest to export the information of the database to I-DEAS. For this purpose, a code was written in VB to convert the material properties in the database to an I-DEAS universal file. The I-DEAS universal dataset number that dealt with the material properties was 1716. The program could successfully export all the desired material properties in the database at different temperature points to I-DEAS material database via the Universal file.

2.6. Other short assignments

Apart from the main tasks assigned during the training, there were several short side-assignments, each of one to two days duration. Below is mentioned a few of them.

2.6.1. Theoretical calculation of stresses on the welds and bolts of the support used for holding a pressure vessel in the workshop

A schematic (not to scale) representation of the problem is shown in the adjacent diagrams.

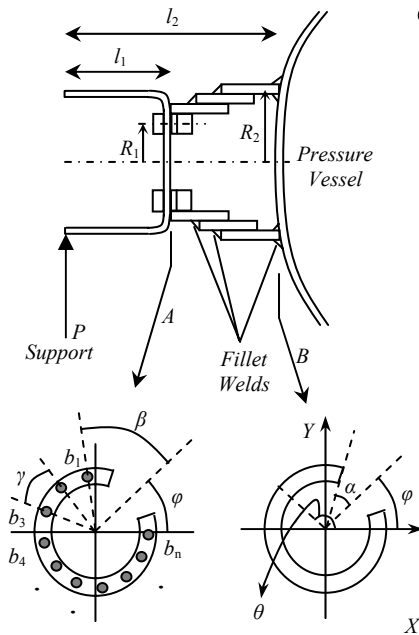


fig - 9

A pressure vessel during its manufacture process needs to be supported at its two ends. The whole load would be carried by the bolts at section A and the welds. As the weld at section B is farthest from the point of support, the stresses can be expected to be most severe there. Hence the stresses in bolts at section A and the weld at section B are to be determined and checked if they are within the stress limits. The figures show that the bolts and welds are not in continuous circles, but there is a gap of 2α for the weld and 2β for the bolts angle in between. The axis of symmetry was assumed to make an angle of ϕ with the horizontal. The angular gap between two bolts was γ .

Although the cross-sections shown in the adjacent figures are not symmetric, for the purpose of simplification it was approximately assumed, when $\phi \approx 0$ and α is small,

$$\text{Longitudinal stresses on the weld} = \frac{M y}{I_{XX}}$$

$$\text{Stress on a bolt} = F_i = k \cdot y_i$$

Where, y = distance of the point on the weld from the horizontal axis = $R_2 \cos \theta$.

y_i = distance of i^{th} bolt from the horizontal axis = $R_1 \cos \theta_i$.

k = a proportionality constant, determined on the basis of equilibrium consideration.

The moment due to the supporting forces at section A = $M_1 = P \cdot l_1$

The moment due to the supporting forces at section B = $M_2 = P \cdot l_2$

The moment of inertia of the weld section about the horizontal axis was determined by integration and was found to be,

$$I_{XX} = R_2^3 t \left[\pi - \alpha + \frac{1}{2} \cos \phi \sin \alpha \right]$$

The angular position of the i^{th} bolt is given by $\theta_i = \phi + \beta + (i-1)\gamma$

And k is determined by equilibrium consideration of moments due to forces F_i , given by the equation,

$$\sum_{\text{all bolts}} F_i y_i = M_1$$

The final expression for the forces on i^{th} bolt is given by,

$$F_i = \frac{M_1 \sin[\varphi + \beta + (i-1)\gamma]}{R_1 \sum_{i=1}^n \sin^2[\varphi + \beta + (i-1)\gamma]}$$

And, the stress on the weld at a point at angular position θ is given by,

$$\sigma(\theta) = \frac{M_2 \sin \theta}{R_2^2 t \left[\pi - \alpha + \frac{1}{2} \cos \varphi \sin \alpha \right]},$$

where t = weld throat thickness.

For the particular problem, the geometric parameters were $\alpha = 35$, $\beta = 45$, $\gamma = 22.5$, $\varphi = 0$, $l_1 = 0.6$, $l_2 = 0.803$, $R_1 = 0.68$, $R_2 = 0.751$, $t = 0.0106$, bolt c.s. area (A) = 0.00102.

The load P was assumed to be half of the weight of the pressure vessel, $P = 343000$.

The allowable bending/tensile strength was taken, $0.6Y = 1.49E+08$.

The allowable shear strength was taken, $0.45Y = 1.12E+08$.

All units in SI system.

All the calculations were performed in MS Excel spreadsheet and the stresses in the bolts and welds were found to be well within the limits. The results are shown in Appendix-K.

[Note: A more accurate and detailed approach would have been to use the following equations,

$$\text{Longitudinal stresses on the weld} = \frac{M(I_{XY}x - I_{YY}y)}{(I_{XY}^2 - I_{YY}I_{XX})}$$

$$\text{Stress on a bolt} = F_i = k_1 \cdot y_i + k_2 \cdot x_i$$

Where, I_{XX} , I_{YY} , and I_{XY} can be determined by integration at section B ,

And k_1 and k_2 determined by both force and moment equilibrium consideration at section A .]

2.6.2. Collection of material test data from job files

This was a short task in which some material test result data were to be collected from job files. This helped in coming to know about how testing of materials are done in the industry, and to get familiarized with job files.

3. Appendices

Appendix-A – Procedure developed for determination of Collapse Load using Ansys

To determine the collapse load, the analyst must first chose a material model with elastic-perfectly plastic property. Such a material is defined by two parameters:

- a. The Modulus of Elasticity of the material in it's elastic region, and
- b. The Yield Stress of the material.

The above data must be available before starting the analysis.

The analysis procedure is described below in a stepwise manner:

- i) Create the model using the tools given in 'Preprocessor > Modeling' menu.
- ii) Define the element type to be used for meshing from 'Preprocessor > Element Type > Add/Edit/Delete > Add...' For an axisymmetric model, typically the element 'PLANE82' under the 'Structural mass > solid' group may be chosen.
 - a. If required, set any options for the element from 'Preprocessor > Element Type > Add/Edit/Delete > Options...' For an axisymmetric model, change the 'Element behavior' to Axisymmetric.
 - b. Also set any required real constant value for the element from 'Preprocessor > Real Constants > Add/Edit/Delete'. This is totally dependent on the element used and the specific problem. For 'PLANE82' there is no need for setting any real constant value.
- iii) Define the material properties from 'Preprocessor > Material Props > Material Model'. Select the 'Material model number 1'.
 - a. Go to 'Structural > Linear > Elastic > Isotropic'. Type the value of Modulus of Elasticity of the material in the 'EX' field and the Poisson's ratio in the 'PRXY' field. Typically, the Poison's ratio has value of 0.3.
 - b. Go to 'Structural > Nonlinear > Inelastic > Rate Independent > Isotropic hardening plasticity > Mises Plasticity > Bilinear' and set the 'Yield Stress' to yield stress of the material, and 'Tangent modulus' to 0.
 - c. Set any other property value if required.
- iv) Go to 'Preprocessor > Meshing > Mesh Attributes > default attributes' and make sure that the correct element, material and real constant set numbers are chosen. Select the size controls from 'Size Cntrls > Smart size (or Manual Size)'. Mesh the model using the meshing tools given under 'Preprocessor > Meshing > Mesh'.
- v) Go to 'Solution > Analysis type > New Analysis'. Select 'Static' and click 'Ok'.
- vi) Open the 'Solution controls' window from 'Solution > Analysis type > Solution Control'. Now ensure that the following are done:
 - a. Under the 'Basic' tab, make sure,
 - i. 'Automatic time stepping' is 'on'.
 - ii. 'Number of substeps' is 20.
 - iii. 'Minimum number of substeps' is 20.
 - iv. 'Maximum number of substeps' is 100.

- v. 'Time at the end of loadstep' is a value between 0.1 and 1. [For a quick and approximate solution use small value (0.1) for this parameter, while for a more accurate solution, typically chose 1.]
- vi. Select 'write every substep' from 'Frequency' under the 'write items to result file' block.
- b. Under the 'Nonlinear' tab, set the following nonlinear options:
 - i. 'Line search' is 'on'
 - ii. 'Equilibrium iteration' is 1000. [Typically this value should be high in order to ensure that Ansys gets enough scope to try to converge the solution. However, the analyst should also keep a careful eye on the convergence graphs during the solution and stop the solution if he/she feels the solution won't converge, just to avoid mere wastage of time by reaching non-convergence after all the 1000 iterations.]
- vii) Chose a set of loads to be applied. Our primary aim is to determine the loads at which the solution is about to diverge. Hence the chosen load set is desirably some value near the collapse load. This may be done by some crude analytical calculation for determination of collapse load set. Let's assume the analytically determined collapse load set is p . Then the actual collapse load will expectedly lie between $0.8p$ and $1.2p$. So starting with $0.8p$ will be a good idea.
Anyway, the analyst may start with any value according to wish, because finally the actual collapse load will be achieved by a sort of iterative method. Let this first load set that is being applied be called P_1 . The consequent load sets applied be P_2, P_3 , etc (see step (x)).
- viii) Apply the load set (P_n for the n^{th} step) using the tools given under 'Solution > Define Loads > Apply' or 'Preprocessor > Loads > Define Loads > Apply'.
- ix) Start the solution from 'Solution > Solve > Current LS'.
- x) Now, two cases may arise:
Case I: The solution converges. This implies that the collapse load is higher than the applied load.
For convenience of further reference, note down the value of this load, and let it be called $P_{\text{just lower}}$ (i.e. load set just lower than the collapse load).
Now an even higher load needs to be applied. If $P_{\text{just higher}}$ already exists, then chose a load somewhere between $P_{\text{just lower}}$ and $P_{\text{just higher}}$, else chose something around $1.2p$. Call it P_{n+1} . for the $(n+1)^{\text{th}}$ step.
Case II: The solution diverges. This implies that the collapse load is lower than the applied load.
For convenience of further reference, note down the value of this load, and let it be called $P_{\text{just higher}}$ (i.e. load set just higher than the collapse load).
Now a lower load needs to be applied. If $P_{\text{just lower}}$ already exists, then chose a load somewhere between $P_{\text{just lower}}$ and $P_{\text{just higher}}$. Call it P_{n+1} . for the $(n+1)^{\text{th}}$ step.
- xi) Delete the already applied load step from 'Solution > Define Loads > Delete' or 'Preprocessor > Loads > Define Loads > Delete'.

- xii) Repeat steps (viii) to (xi) until sufficiently close values of $P_{just\ lower}$ and $P_{just\ higher}$ are obtained.
- xiii) The load set $P_{just\ lower}$ approximately gives the collapse load, two-third of which gives the limit load.

Appendix-B – Procedure developed for determining whether shakedown occurs at a particular Load set using Ansys

To perform shake down analysis directly using Ansys, without using Melan's Theorem, one may approach in similar iterative approach as described in Appendix-A. That is, determine if shake down occurs at a particular load or not and then go on changing the loads in a trial and error fashion to finally determine the maximum load at which shakedown occurs.

But the problem with this is that the shakedown analysis using Ansys is a bit time consuming process as it involves several loading-unloading cycles. Hence the following procedure may at most be used efficiently to predict if shakedown will take place under a particular given load, but not to determine the shakedown load.

To perform a shakedown analysis, several cycles of loading and unloading need to be applied on the structure. Let the loads applied during loading be defined by a load set P . The loads applied during unloading will definitely be zero. We will perform four such cycles and see if shakedown occurs by then.

Peruse the following steps to determine if shakedown occurs at the particular load set:

- i) Create the model, define element type, define material, and mesh the model as described in steps (i) to (iv) of Appendix-A.
- ii) Go to 'Solution > Analysis type > New Analysis'. Select 'Static' and click 'Ok'.

[Note:

By opting to perform a Static analysis we have eliminated any possible involvement of the inertia forces due to mass matrix. This will hence imply that the frequency of the loading-unloading cycles is very low (i.e. the cycles are very slow). The value of Δt (see next step) hence will have no effect on the solution results. The analyst may chose any convenient value for Δt .

But if the analyst is interested in including the effects of cycle frequency and the inertia forces, he/she need to perform a transient analysis. To do this, select 'Transient' from 'Solution > Analysis type > New Analysis'. Then select 'Full' from the window that pops up and click 'Ok'. The remaining procedure is same as that follows. However the time required by Ansys to perform the solution will be much higher.]

- iii) Define the 'Solution controls' exactly as described in (vi) of Appendix-A. Let the value chosen for the parameter 'time at the end of load step' be Δt . This value (Δt) will be referred later to define the load steps.
- iv) Apply the load set P from 'Solution > Define Loads > Apply' or 'Preprocessor > Loads > Define Loads > Apply'.
- v) Go to 'Solution > Load Step Option > Write LS file'. Enter '1' as load step file number. Click 'Ok'.

Note: By this process, the first load step has been written, which is the loading half of the first cycle. The time at the end of this first half of the first cycle was defined under ‘Solution controls’ as Δt . As the same load needs to be applied in the first halves of 2nd, 3rd and 4th cycles, we can write the load step file numbers 3, 5 and 7 with the same load set, just by changing the ‘time at the end of load step’ for each. So we do in the following step.

- vi) In ‘Solution > Analysis type > Solution Control’, change ‘time at the end of load step’ to $3 \cdot \Delta t$. Go to ‘Solution > Load Step Option > Write LS file’. Enter ‘3’ as load step file number. Click ‘Ok’.
- vii) Similarly write the Load step files number 5 and 7, with ‘time at the end of load step’ as $5 \cdot \Delta t$ and $7 \cdot \Delta t$ respectively.
- viii) Next remove all the loads (but not the DOF constrains) from ‘Solution > Define Loads > Delete’ or ‘Preprocessor > Loads > Define Loads > Delete’.
- ix) Apply zero load (or a very small load compared to \mathbf{P} for numerical stability).
- x) Save the Load Step files 2, 4, 6 and 8 with ‘time at the end of load step’ values $2 \cdot \Delta t$, $4 \cdot \Delta t$, $6 \cdot \Delta t$ and $8 \cdot \Delta t$ respectively.
- xi) Go to ‘Solution > Solve > From LS Files’. Set ‘Starting LS file number’ as 1, ‘Ending LS file number’ as 8, ‘File number increment’ as 1, and click ‘Ok’ to start the solution.
- xii) Once the solution is done, to view the results use the following procedures:
 - o Viewing Plastic Strain vs. time plot at a particular point in the structure:
 - a. Open the ‘Time History Variables’ window by clicking on the ‘TimeHist PostPro’ menu or by using ‘TimeHist PostPro > Variable Viewer’.
 - b. Click on the green ‘+’ button at the upper left corner of the window.
 - c. In the ‘Result Item’ double-click on item to open it’s sub contents. Hence select ‘Nodal Solution > Plastic Strain > von Mises plastic strain’. Click ‘Ok’
 - d. Ansys will prompt to select a node from the model. Click on a node at which investigation regarding occurrence of shakedown is sought. Click ‘Ok’.
 - e. A new item will be added to the list in the ‘Time History Variables’ window. Select that item and click on the ‘graph data’ button at the left side of the toolbar. This will plot the graph of Plastic Strain with time at that node.

Typically, for a point where plastic strain takes place at the very first cycle, and subsequent shakedown has taken place, the graph will look as follows:

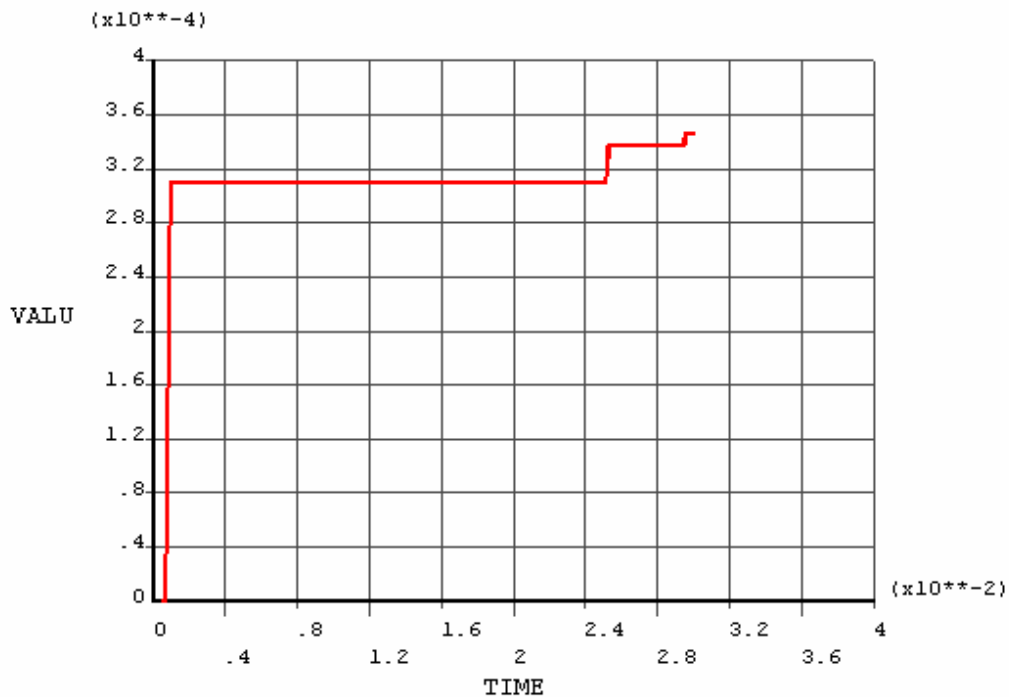


fig - 10

- Animate results over time:
 - a. Read the first set of results from 'General PostProc > Read Results > First Set'.
 - b. In the utility menu (the text menu at the top of the window), 'PlotCtrls > Animate > Over Time...'
 - c. Change number of animation frames to 50. Select 'Load Step Range' and type 1 to 8 in the text boxes below.
 - d. Select the item for which contour animation is to be done and click 'Ok'.

The von Mises Plastic Strain vs. time plots at the different nodes are of special importance for prediction regarding shakedown. Shakedown will be said to have taken place at a node if the value of von Mises plastic strain at the node approaches a steady value with time. But if it goes on raising steadily or increasingly, then ratcheting may be said to have taken place. The other case that may happen is that the von Mises plastic strain remains zero throughout. In that case it may be concluded that the material at that node hasn't crossed the yield point at all.

To draw conclusion about the results, the analyst need to investigate the Plastic Strain vs. time plot at several nodes in the structure, especially the critical points where the stress intensities or the equivalent stresses are expected to be high. If at all the nodes it is found that shakedown has taken place within these 4 cycles, then it may be concluded that shakedown has taken place for the structure under that load set. However, occurrence of ratcheting at some local bounded regions may be allowed to some extent.

Appendix-C – An attempt of calculations for Melan’s theorem using Element Tables in Ansys

It is possible to perform all the calculations for Melan’s theorem using Ansys element tables, but at a cost of substantial amount of labor. Hence the analyst may choose to implement the procedure mentioned in Appendix-D instead of the one mentioned below. However, for using the element tables, the analyst may try to proceed in the following steps:

- i) Perform the analysis using elastic-perfectly plastic material model as described in Appendix-A till the near collapse load is reached.
- ii) Apply the last obtained $P_{just\ lower}$ and perform the solution once again. The stress field now residing in the result is the $(\sigma_{ij})_{ep,l}$.
- iii) Go to ‘General PostProc > Element Table > Define Table’.
- iv) Click the ‘Add...’ button and select the X component of stress (SX) from the list. Type ‘SX1’ in the ‘User label for item’ text box. Click ‘Ok’ to add the item.
- v) Similarly add the stress components SY , SZ , SXY , SYZ and SZX with user labels SY1, SZ1, SXY1, SYZ1 and SZX1 respectively. [It may be noted here that for axisymmetric models, as SYZ and SZX will always be zero, they needn’t be included.]
- vi) Now as the $(\sigma_{ij})_{ep,l}$ is stored, we need to create the $(\sigma_{ij})_{el,l}$ field. To do this, go to ‘Preprocessor > Material Props > Material Model’ and open the contents of ‘Material model number 1’ by double-clicking it. Select ‘Bilinear Isotropic’ and delete it using ‘Edit > Delete’. Delete ‘Bilinear Isotropic’ for any other material present.
- vii) Solve the model once again with the same load set $P_{just\ lower}$.
- viii) Follow steps (iii) to (v) to create the element tables of the stress components SX , SY , SZ , SXY , SYZ and SZX with user labels SX2, SY2, SZ2, SXY2, SYZ2 and SZX2 respectively.
- ix) Now that we have got both $(\sigma_{ij})_{ep,l}$ and $(\sigma_{ij})_{el,l}$, we need to perform the required calculations with them. The main disadvantage of operations with element tables is that Ansys allows a single operation between two tables at a time. So the analyst needs to break up his/her calculations into several single-step calculations to perform the whole calculation.
- x) To find the self-equilibrating stress field $(\sigma_{ij})_{res}$, use ‘General Postproc > Element Table > Add Items’. Set,
 - o ‘User label for result’ as SX3,
 - o 1st factor as 1,
 - o 1st element table as SX1,
 - o 2nd factor as –1,
 - o 2nd element table item as SX2, and
 - o Constant as 0.

Click ‘Ok’.

Go to ‘General PostProc > Element Table > Define Table’ to check if ‘SX3’ has been added.

- xi) Similarly, create SY3, SZ3, SXY3, SYZ3 and SZX3.
- xii) As it has already been made evident, in the previous few steps we subtracted each components of $(\sigma_{ij})_{el,l}$ from $(\sigma_{ij})_{ep,l}$ to obtain $(\sigma_{ij})_{res}$. It is also possible to perform multiplication, find absolute value, determine maximum or minimum among different table items at each node, or exponentiate (square,

cube, square-root, etc) the items in the table using the tools under ‘General Postproc > Element Table’.

- xiii) Next step is to find the principle stress components of $(\sigma_{ij})_{res}$. The calculations involved in doing so are rather tedious and doing so using Ansys element tables would require lots of steps. The detailed equations involved for the calculations can be found in any standard text book on advanced solid mechanics. The calculations involved in determining the principal stresses from the component stresses are described in step ii. of the pseudo-algorithm in Appendix-E. However, the problem is a bit less severe when the model is plane stress, where SZ , SYZ & SZX are all zero.

However in any case, the calculations involved are quite tedious to be performed one step at a time using the element table operations. Hence it may be advised to follow the method described in Appendix-D.

Appendix – D – Calculations for Melan’s theorem using a computational code to read the stress fields as obtained from an Ansys run

The easiest and most flexible way of dealing with the components of $(\sigma_{ij})_{ep,l}$ and $(\sigma_{ij})_{le,l}$ is to export these stress fields from Ansys and read them using a program written in any convenient programming language. Calculating α and β is then just a matter of manipulation with the components.

Below is described the procedure of exporting the stress fields from Ansys to files in ASCII text format which can be read with some computer program:

- i) Perform the analysis using elastic-perfectly plastic material model as described in Appendix-A till the near collapse load is reached.
- ii) Apply the last obtained $P_{just\ lower}$ and perform the solution once again. The stress field now residing in the result is the $(\sigma_{ij})_{ep,l}$.
- iii) In the utility menu, go to ‘List > Results > Nodal Solution’. Select ‘Stress’ from the left list, and ‘components SCOMP’ from the right. Click ‘Ok’ to list the stress components. In the list window, use ‘File > Save as...’ to save the list in ‘.lis’ file format. This list file may now be easily read using a program.
- iv) Now as the $(\sigma_{ij})_{ep,l}$ is stored in a file, we need to create the $(\sigma_{ij})_{le,l}$ field. To do this, go to ‘Preprocessor > Material Props > Material Model’ and open the contents of ‘Material model number 1’ by double-clicking it. Select ‘Bilinear Isotropic’ and delete it using ‘Edit > Delete’. Delete ‘Bilinear Isotropic’ for any other material present.
- v) Solve the model once again with the same load set $P_{just\ lower}$.
- vi) Once again save the stress components’ list as a ‘.lis’ file as described in step (iii).
- vii) Refer to Appendix-E for working with the two stress fields obtained using a computer programming language.

Appendix-E – Pseudo-code of the VB program developed for determination of α and β from the stress fields using Melan's theorem

Using Appendix-D both the stress tensors $(\sigma_{ij})_{ep,l}$ and $(\sigma_{ij})_{le,l}$ are obtained at each node of the model. Next we need to have a code for reading the stress fields and operating with them to obtain α and β . Such a code in MS Visual Basic has already been developed for this purpose. The analyst may make use of that.

The pseudo-algorithm for the code is described below:

- i. Calculate $(\sigma_{ij})_{res}$ by subtracting each component in $(\sigma_{ij})_{le,l}$ from $(\sigma_{ij})_{ep,l}$ for all the nodes.
- ii. Find the Equivalent Stress (for von Mises's criteria) or the Stress intensity (for Tresca's criteria) at each node. For calculating these, one need to first determine the principle stresses from the component stresses of $(\sigma_{ij})_{res}$ at each node. The following sets of equation may be helpful:

If $\sigma_x, \sigma_y, \sigma_z, \sigma_{xy}, \sigma_{yz},$ and σ_{zx} are the stress components, then define the stress invariants as,

$$\begin{aligned} J_1 &= \sigma_x + \sigma_y + \sigma_z \\ J_2 &= \sigma_x \cdot \sigma_y + \sigma_y \cdot \sigma_z + \sigma_z \cdot \sigma_x - \sigma_{xy}^2 - \sigma_{yz}^2 - \sigma_{zx}^2 \\ J_3 &= \sigma_x \cdot \sigma_y \cdot \sigma_z + 2 \cdot \sigma_{xy} \cdot \sigma_{yz} \cdot \sigma_{zx} - \sigma_z \cdot \sigma_{xy}^2 - \sigma_x \cdot \sigma_{yz}^2 - \sigma_y \cdot \sigma_{zx}^2 \end{aligned}$$

Let,

$$\begin{aligned} G &= \frac{J_1 J_2}{3} - \frac{2J_1^3}{27} - J_3 \\ H &= -J_2 + \frac{J_1^2}{3} \end{aligned}$$

Determine θ from the equation,

$$\cos 3\theta = \frac{G}{2} \left(\frac{H}{3} \right)^{-3/2}$$

Let $\frac{\gamma}{3}, \frac{\gamma + 2\pi}{3}$ and $\frac{\gamma + 4\pi}{3}$ be the three solutions of θ . Then the principal stresses are given by the equation,

$$\sigma = \frac{J_1}{3} + 2\sqrt{\frac{H}{3}} \cos \theta$$

from which the three principal stresses, σ_1, σ_2 and σ_3 are obtained for the three values of θ .

Once the principal stresses are obtained,

Equivalent Stress (for von Mises's criteria) is given by,

$$\sigma_{eq} = \sqrt{\frac{1}{2} \{ (\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 \}}$$

and Stress intensity (for Tresca's criteria) is given by,

$$\sigma_{int} = \max(|\sigma_1 - \sigma_2|, |\sigma_2 - \sigma_3|, |\sigma_3 - \sigma_1|)$$

- iii. Now check at each node if the Equivalent Stress (or the Stress Intensity) satisfies the yield condition. Hence two cases may arise:

Case I : Yield condition is satisfied at all nodes:

Then $\beta = 1$ and $\alpha = 1$. Hence, shakedown load is equal to the collapse load. Skip all the following steps.

Case II: Yield condition is not satisfied at some point.

Determine the node where the Equivalent Stress (or the Stress Intensity) is the maximum. Then,

$$\beta = \text{yield stress} / \text{this maximum stress.}$$

iv. When $\beta \neq 1$, at each node do the following:

a. Calculate $(\sigma_{ij})_{res,co}$ multiplying each component of $(\sigma_{ij})_{res}$ by β .

$$(\sigma_{ij})_{res,co} = \beta \cdot (\sigma_{ij})_{res}$$

b. Assume values of α' from 0 to 1 at steps $\Delta\alpha'$ (may be done in a 'For loop') and check if the following stress tensor satisfies yield condition (for this, the equations of step ii. may again be used):

$$(\sigma_{ij})' = (\sigma_{ij})_{res,co} + \alpha' \cdot (\sigma_{ij})_{le,l}$$

c. Hence at each node a permissible range of α' will be obtained for which $(\sigma_{ij})'$ satisfies the yield condition. Let this range be represented by (α'_1, α'_2) .

v. Now that we have obtained a range (α'_1, α'_2) for each node, we find the common range of all these by finding their intersection. The range hence obtained is the permissible range of α' . The highest value in this range (i.e. the upper limit) gives the required value of α .

vi. Hence, the shakedown load is given by,

$$\mathbf{P}_{SD} = \alpha \cdot \mathbf{P}_{lim}$$

Appendix-F – *Verification of the method for finding Collapse Load (Appendix-A) and Shakedown Load (Appendix-B, D and E) by re-performing a Finite Element Analysis*

For the purpose of validation of the developed methods described in Appendix-A, D and E, a sample problem consisting of shell and plate was taken for which collapse load and the parameters α and β from Melan's theorem has already been determined using some alternative routes in the '*Design by Analysis CD Manual*' issued by European Pressure Equipment Research Council, University of Strathclyde. The results obtained by these methods were in satisfactory agreement with those of the manual.

The following figures show the meshed model:

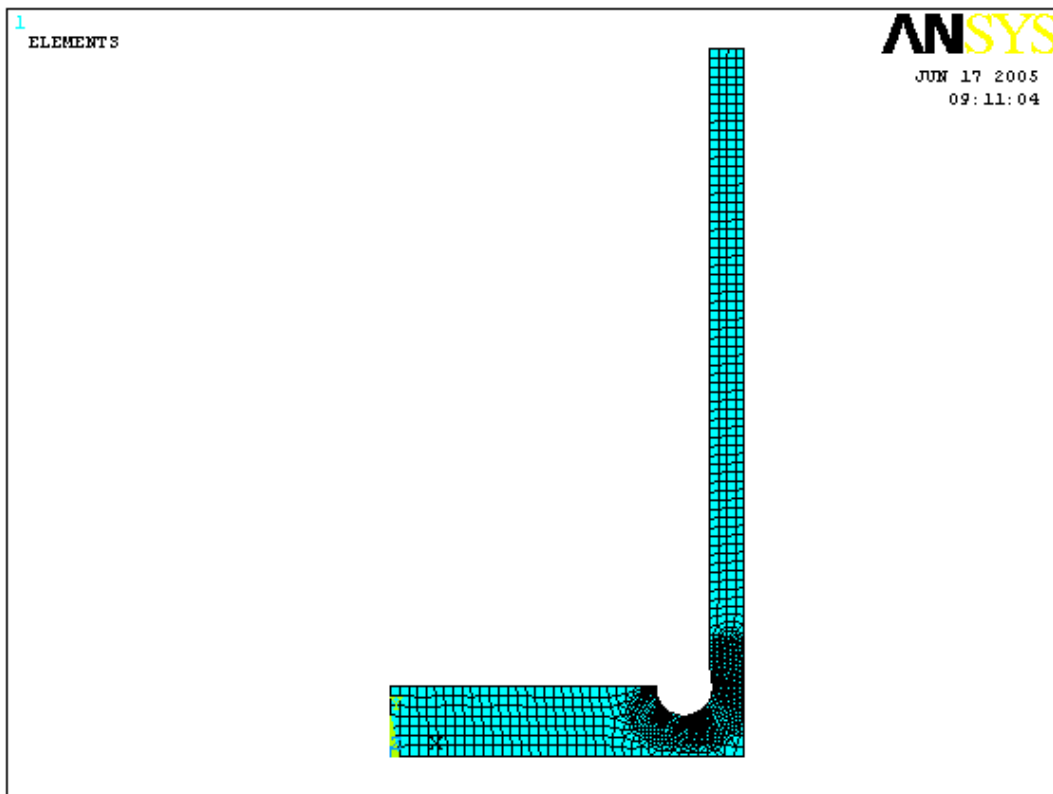
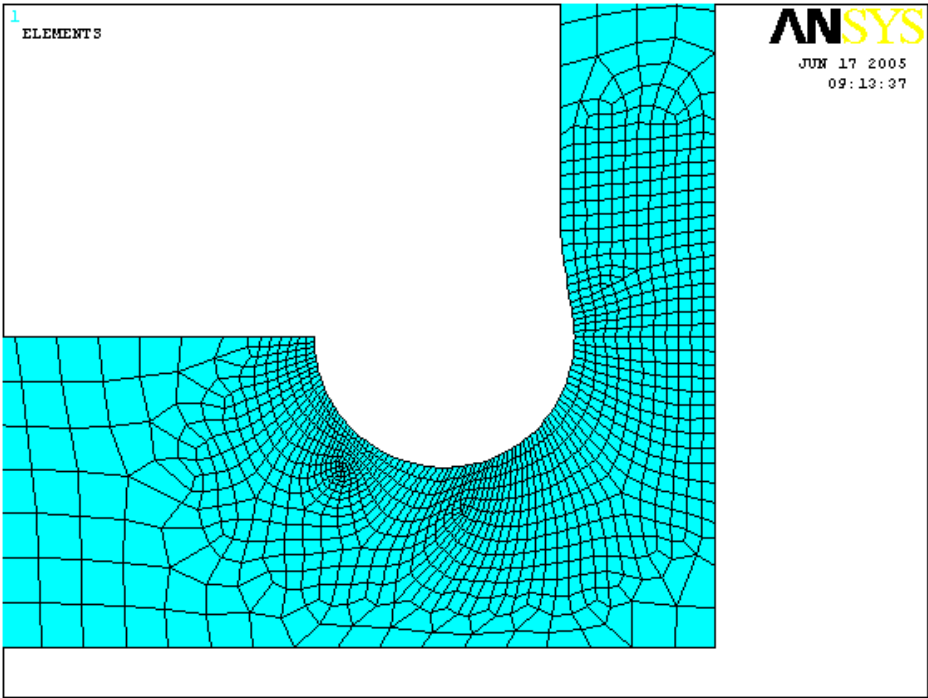
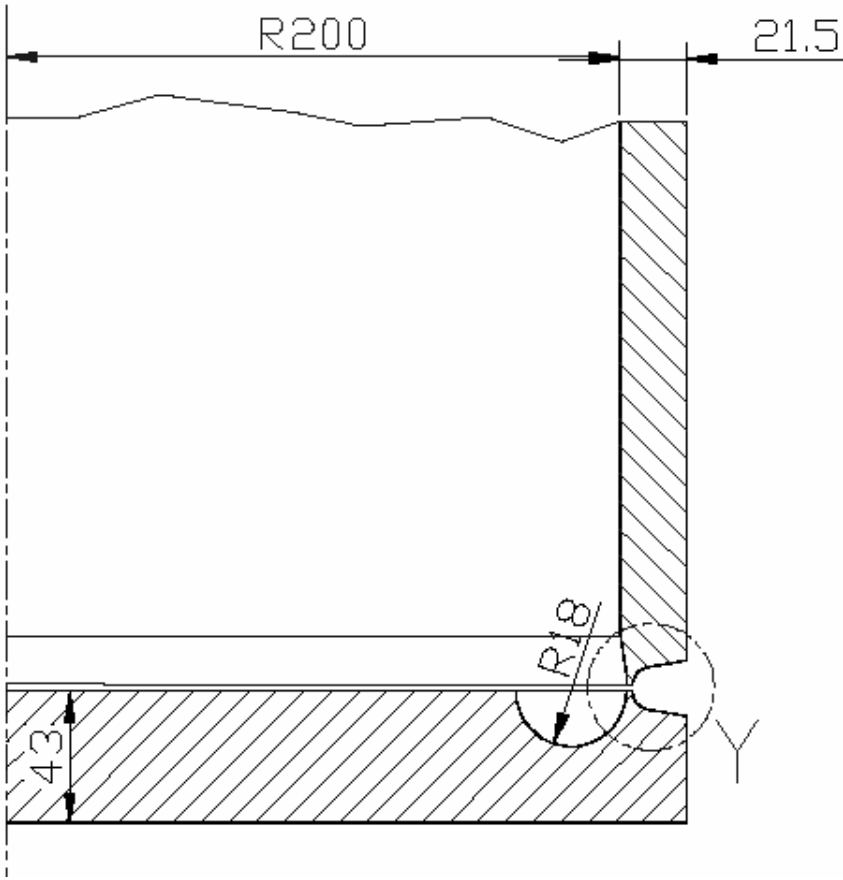


fig - 11



→ fig - 12

The dimensions used are showed in the following figure:



→ fig - 13

The total height of the model is 443mm.

Element type: As used in the original problem, here the element used was 4-node, 2-D axisymmetric solid PLANE42. The number of elements generated in the present case was 1743 using free meshing tool.

Boundary Conditions: Zero vertical displacement was applied on the undisturbed end of the shell (the horizontal line at the upper end of the model). Symmetry boundary conditions applied on the center of the flat shell (the leftmost vertical line).

Loads: A uniform pressure P was applied on the inner surface of the model, i.e. the upper surface of the plate, the left surface of the shell and the inner surface of the circle.

Material Properties: The modulus of elasticity of the material in the whole structure is $E = 212\text{Gpa}$. The yield stresses were chosen to be 255Mpa for the shell and 245Mpa for the plate materials.

Results obtained:

- i. By following the method described in Appendix-A, the collapse load obtained was approximately 21.5MPa .
- ii. As described in 4.2., the values of α and β obtained using the MS Visual Basic program were $\beta = 0.4445$ and $\alpha = 0.6217$.

[Notes A:

- a. The value of $\Delta\alpha'$ used was 0.0001 ,
- b. As it was found from the plastic strain contour plot at load 21.5MPa that the plastic collapse begins in the plate, the value of yield stress used for calculations for Melan's theorem in the code was assumed to be 245Mpa . This was done for simplification of the code and the procedure. But a more rigorous and correct approach would have been to use different values for yield stress at different nodes depending on in which part of the model the node lies.]
- iii. Hence the shakedown load is given by, $P_{SD} = 0.6217 \times 21.5\text{Mpa} = 13.37\text{MPa}$.

Comparison with *DBA CD Manual*:

The collapse load used in the *DBA CD Manual* for calculation of shakedown load is 21.43Mpa , and The values of α and β obtained were $\alpha = 0.621$ and $\beta = 0.444$. The shakedown load was hence $P_{SD} = 13.3\text{Mpa}$.

Hence it can be concluded that the results obtained are in quite agreement with those of the *DBA CD Manual*.

The following figure show the plastic strain at load $P = 21.5\text{MPa}$ at a region of high stress (near the neck of the structure):

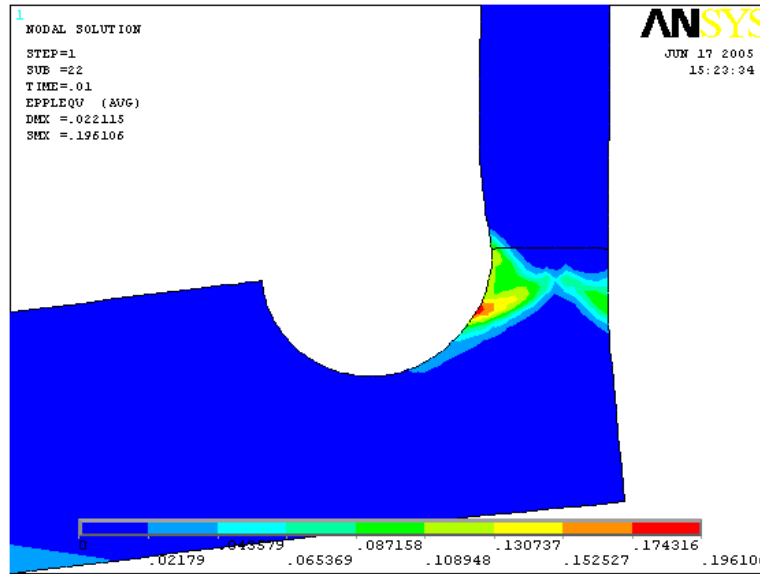


fig - 14

Determining if shakedown occurs at the shakedown load

Now as the shakedown load was obtained using Melan’s theorem, the procedure described in Appendix-B was used on the same model as described above to determine if shakedown occurs at this load of $P_{SD}=13.37\text{Mpa}$.

The following plot shows the plastic strain vs. time curve a node near the high-stressed neck of the structure. For the other nodes either the plastic strain remained zero throughout, or shakedown had taken place just after the first cycle similar to the plot shown.

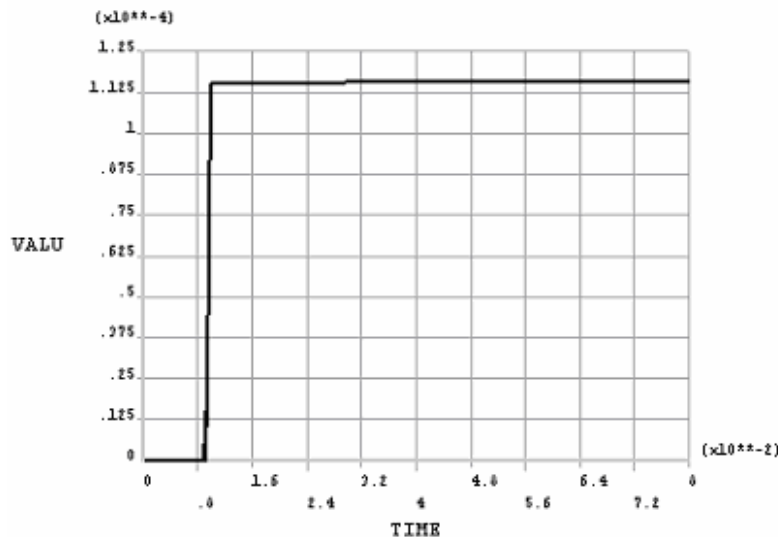


fig –15 Plastic strain vs. Time plot at node no. 319 – a point near the high stressed region

Hence it may be concluded that shakedown takes place at the determined load of 13.37Mpa.

Appendix-G – Elements used, Material Properties, Boundary conditions applied and other details for the tube to tubesheet junction simulation of the boiler

1	Component	TUBE TO TUBESHEET JOINT
2	Analysis type	COUPLED FIELD (STATIC AND THERMAL)
3	Software used	Ansys 8.0
4	Element type	1st order Coupled-field Axisymmetric Elements PLANE13, Mechanical Contact Element CONTAC48, Thermal Contact and Target Elements CONTA172 and TARGET169
5	No of elements	3687 PLANE13, 1558 CONTAC48, 510 CONTA172, 510 TARGET 169
6	No of nodes	4923

7 Design data :

Design code	ASME SECT. VIII, DIV 2, ED.2001, AD.2003. & TEMA LATEST ED.
Corrosion allowance	SHELL SIDE - 1MM, TUBE SIDE - 1 MM (NIL FOR TUBES)

8 Material properties :

No	Part	Material	Design Temp deg C	All. Stress, S_m , kgf/(mm ²)	Mod of elasticity, E, kgf/(mm ²)	Thermal cond. Kcal/(s-mm-C)	Coeff of thermal exp. [mm/mm/C]	Yield Strength, Y, kgf/(mm ²)
1	TUBESHEET	SA 336 GR F22 CL3	480	12.20	19290.25	8.6416E-06	1.35E-05	22.8
2	TUBE	SA 213 GR T22	380	12.58	19290.25	8.6416E-06	1.35E-05	18.89
3	LAS PORTION OF FERRULE	SA 213 GR T22	480	9.73	18240.52	8.2575E-06	1.42E-05	18.03
4	FERRULE	INCONEL 600 (SB 167 UNS NO6600)	480	11.29	18768.55	5.0824E-06	1.46E-05	19.24

* Allowable stress as per ASME Section VIII, Div-1

9 DOF constrains and Mechanical boundary conditions:

Pressure:

Location	Pressure [kgf/(mm ²)]
Tube Side	2.45
Shell Side	1.30
Upper end of tube (axial loading)	-1.319

DOF constrains:

Location and DOF name	Value
Lower surface of tubesheet - vertical displacement (UY)	0

10 Thermal boundary conditions:

Convection Boundary conditions:

Location	Conv. Heat transfer coeff. [kcal/(s-C-mm ²)]	Temperature (deg C)
Below tubesheet	9.73E-07	455
Inside tube	7.55E-07	455
Outside tube (Shell side)	5.56E-06	324

Heat flux boundary condition:

Location	Heat flux
On the extreme edge of the portion of tubesheet under consideration	0

Appendix-H – Results obtained from the tube to tubesheet junction simulation of the boiler

The stress values were obtained from the results and hence the stress intensity contour plots were obtained. Moreover the stresses were linearized at some critical sections (refer to *fig 5*). Hence the membrane stress intensities and bending stress intensities were obtained at those sections.

The stress contour plots and the stress values obtained after linearization at the sections cannot be shown in this report because of proprietary reasons. However the temperature contour plots at two positions in the model are shown in the following page.

12 Temperature Contour Plots:

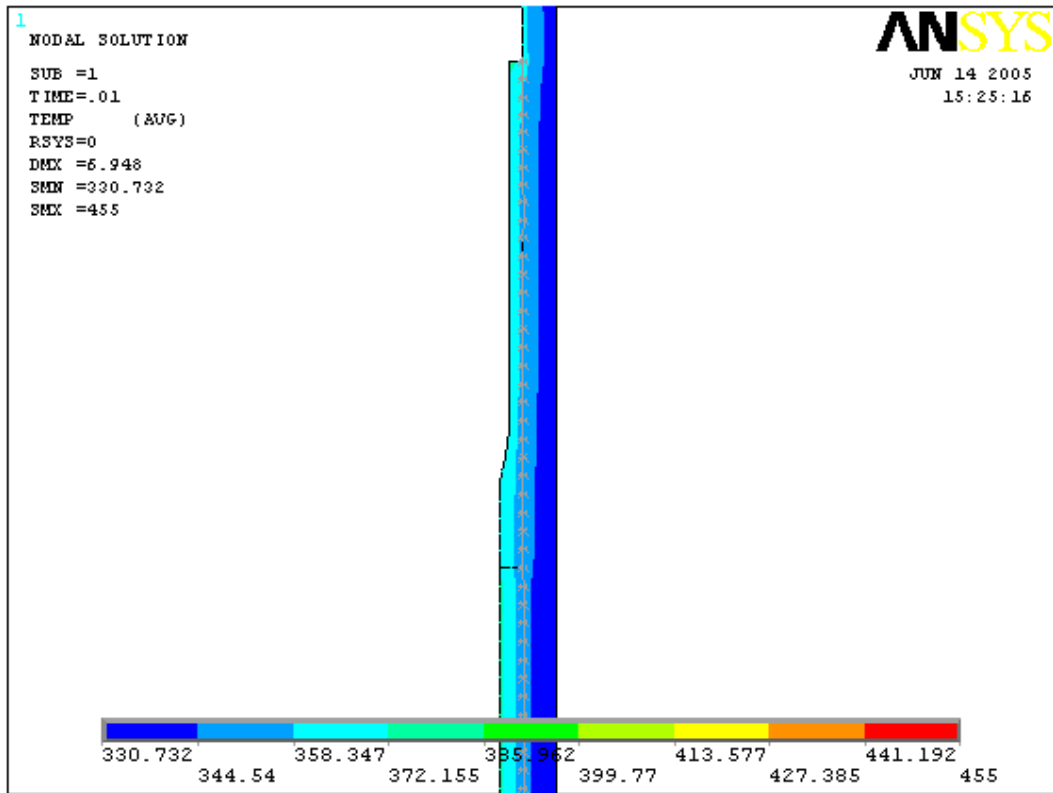


fig - 16a

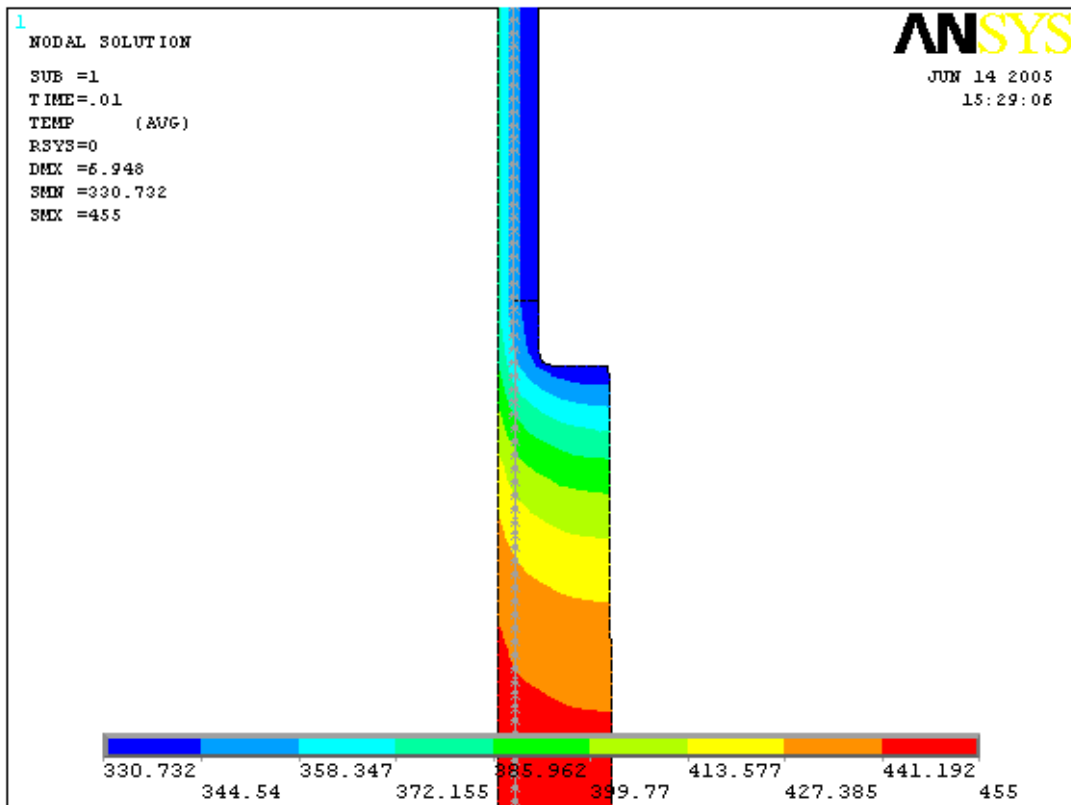


fig - 16b

Appendix-I – *Documentation made for Working manually with the MS Access Material Database*

I.1. Retrieving a Value from the database

To extract a value from the database proceed through the following steps:

- i) Open the ‘Materials’ database and find out the material name of which the property value is sought.
- ii) Note the ID number that is present in a property field, the value of which is being sought.
- iii) Open the ‘Data_’ table corresponding to that property.
- iv) Scroll down and search for the set of records for which the ‘ID’ fields contain the same ID number as that noted from the ‘Materials’ table in step ii) (i.e. the property data set). All such records’ Field 1 (temperature) and Field 2 (property value) gives the property values at different temperature points.

I.2. Editing the already existing Values

To edit a property value of the material, or the temperature entries, just go to the corresponding position as described in I.1. and edit as required.

I.3. Adding values to an already existing material’s property

To add a temperature and value point for an already existing material in the database, do the following:

- i) Open the ‘Materials’ database and find out the material name of which the property value is sought.
- ii) Note the ID number that is present in a property field, the value of which is being sought.
- iii) Open the ‘Data_’ table corresponding to that property.
- iv) Add a record at the end of the ‘Data_’ table with Field 0 equal to the ID number noted in step ii) and Field 1 and Field 2 equal to the temperature and property value to be added. Note that you must use 8F as temperature units and the unit system in accordance to ASME Sec II Part D for the property values.
- v) Sort the database first with respect to Temperature, next with respect to ID.

I.4. Adding a new material to the database

To add a new material to the database, proceed as follows:

- i) Open the ‘Materials’ database and add a new record at the end of the database with Field 0 (Material Name) as the name of the material.
- ii) For a particular property, check whether the property data set intended to be inputted already exists in the corresponding ‘Data_’ property table. [There is no definite procedure for doing this. One may check it from knowledge of the material groupings made in the ASME code, or may go through all the property data sets in the ‘Data_’ table and find out if the dataset already exists]

- iii) Hence, two cases may arise now:
Case I: The property data set for the material's particular property already exists:
In such a case, just put the ID number of the property data set into the appropriate field beside the material's name in the 'Materials' table.
Case II: A new property data set needs to be added for the particular property:
Pursue the following steps:
- i) Open the 'Data_' property table.
 - ii) Scroll down the records to find what is the maximum number that has been used as ID number in the table. Hence determine the next available ID.
 - iii) Open the corresponding 'ID_' property table and make sure that the ID number exists. If not, add it.
 - iv) Open the 'Materials' table and put the foresaid ID number in the appropriate field beside the material.
 - v) Go back to the 'Data_' table and keep adding records with the same ID number in the 'ID' field, and temperature & property values in the fields 1 & 2 respectively.

Appendix-J -- *Documentation made for Working with the forms developed in the MS Access Materials database*

J.1. Converting the MS Access database to I-DEAS Universal file of material properties

The database can be converted to I-DEAS 11NX universal file. The steps involved are:

- i) The file 'IDEAS_material_format.dat' and 'IDEAS_materials_header.dat' should present in the same folder as the access database file 'Material Properties.mdb'. If they are not present, copy them into the folder of the database.
- ii) Double-click to open the 'Export database to IDEAS UNV file' form. Select the path and file name for exporting the database using the 'Browse...' button.
- iii) Click 'Export database' to export database.

J.2. Finding and Interpolating values using different units

For seeking a particular property value at a particular temperature by linear interpolation upon the database values, perform the followings:

- i) Open the 'Seek/Interpolate' form.
- ii) Select the material name and the property name from the respective drop-down lists.
- iii) Select the desired units for temperature and the property value.
- iv) Click the 'Seek' button.

Notes B:

- o All the values displayed in the form are in the units selected by the user.
- o The values displayed do not get updated unless the 'Seek' button is clicked.

- The user may wish to open the 'Materials' table or the property 'Data_' table directly at the positions from where the last values have been sought, by clicking the 'Open 'Materials' Table' and 'Open Property Value database Table' buttons respectively.

J.3. *Listing, adding, editing and deleting records of the existing database*

The 'List/Edit/Add' form gives the user the ultimate authority on the database in a user-friendly manner. Though care has been taken to ensure that the form works properly under all circumstances and doesn't crash, it is always advisable to create a backup of the database before editing/modifying it with this form.

To add, edit and/or delete records from a property data set using this form, pursue the following steps:

- Open the 'List/Edit/Add' form.
- Select a material from the drop-down list of which a property data set is intended to be edited/modified.
- Select the material property from the drop-down list.
- Now note:
 - This has loaded the temperature & value points of the property data set assigned to the selected material's selected property, which will hence be displayed in the list.
 - In the upper half of the right side of the list, the user may see the ID of the currently loaded property data set.
- Click on a record in the list. This will transfer its value to the 'Temperature' and 'Property Value' text boxes on the right side of the list.
- Make change to the values in the text box or type completely new values and click the 'Add Record' or 'Update Record' Button. Click 'Delete Record' to delete the selected record.
- After the desired changes have been made, click the 'Update database with current list' button at the bottom of the form.
- A dialog box will pop up asking for confirmation whether to overwrite/update. Click 'yes' to save the changes.

Notes C:

- 'Add Record' will add a new temperature and property value point. The record will be added at the position of currently selected record in the list. However the position of a record is of not much importance other than ease of viewing. 'Update Record' will update the presently selected record with the values in the text box. 'Delete Record' will delete the presently selected record.
- The user may wish to view/enter values in different unit systems by choosing the appropriate units from the 'Material Property unit' and 'Temperature unit' drop-down lists in the upper right side of the form.
- The user may choose to work in any unit system throughout without worrying what the unit system of the actual database is. The program takes total care of all the unit conversions.
- The user may click the 'Sort List' button to view the list sorted in ascending order of temperature.

- The user must click the 'Update database with current list' button to save the changes made to the property data set, before he/she can switch to some other material or property. Once switched to other material or property, the unsaved changes made to the already load set will be lost.
- Though, by default the ID number of the data set to which the currently loaded list to be saved will remain the original ID of the Set, the user may opt to save the list to a data set with a different ID by typing the desired ID number in the 'Save this property with ID set number' text box. The ID number should be a positive integer less than 1000. Type 'NEW' or '0' to create a new data set with the lowest unoccupied ID number.

J.4. *Adding a new material to the database and creating, editing it's property values*

It is also possible to add a new material to the database using the 'List/Edit/Add' form. To do so, the steps involved are:

- i) In the 'List/Edit/Add' form type the name of the new material to be added in the 'Material Name' box.
- ii) Type the composition of the material in the 'Composition' box. Or select an already existing composition from the drop-down list. [For typing new material composition, use the standard format as used for the other materials.]
- iii) Click the 'Add Material' button to add the material to the database.
- iv) Do not change the material name or close the form unless the following points have been considered. Please read on ...

Depending on the composition of the material the program may assign some default values to the ID numbers for the different properties of the material if the composition already existed in the database. The user may however opt to change the property set ID numbers for the material to some other value, or may create a new property data set by entering new values. These are to be done in the following steps:

- i) Select the property whose ID/values is to be changed/modified.

Note: If some default ID number had been assigned to the property by the program, the corresponding data set will be displayed in the list and the ID number will be displayed beside the list. Else the list will remain blank and the ID number will be displayed as 0.
- ii) If required, to load an already existing property data set to the list, type the ID number of the set into the 'Load an already existing property Set' text box and click 'Load set to list' button. This will load the set to the list.
- iii) Now the user may edit the list as described in steps v) and vi) of J.3. New temperature and value points may be added as desired.

Tips: To Add several temperature and value points to the list, the following key combinations may be useful:

- a. Use the 'Space bar', 'up arrow' or 'down arrow' keys to toggle between the 'Temperature' and 'Property Value' text boxes.
- b. Use the Enter key (or Carriage Return) to add the presently typed temperature and property value to the list.

- iv) Once the list contains the desired values, click the ‘Update database with current list’ button to add the list to the database as a property data set.

Note: The ‘Save this property data set with ID number’ text box indicates the ID of the data set to which the values will be saved. The user may chose to use a different number or may type ‘NEW’ or ‘0’ to create a new data set with the lowest unoccupied ID number. However, if unsure, the user may leave this value as it is.

- iv) A data set addition/modification confirmation message box will pop-up. Click ‘Yes’ to add/modify.

Notes: All the notes mentioned under ‘Notes C’ will also apply here.

J.5.2. *Restrictions of the ‘List/Edit/Add’ form*

The ‘List/Edit/Add’ form has some restrictions. They are primarily to prevent accidental loss/damage of the database. The restrictions are:

- i. The form cannot be used to delete a material from the database.
- ii. The form cannot be used to delete a data set. However it can be used to modify it’s records, add to the records or delete records one by one.
- iii. The form cannot be used to modify the name, composition or the IDs assigned to the properties of an already existing material. Any of these parameters/values are to be assigned at the time the material has been created.

J.5.1. *Restrictions on the database for the code to work properly (Limitations)*

The following constrains should be maintained on the database to ensure that the forms work properly.

- i. The ID number of the property data sets must be a positive integer less than 1000.
- ii. There shouldn’t be more than 100 records in each data set.
- iii. The standard format should be used for representation of material composition.

If the above restrictions are to be relaxed, the corresponding changes must be made to the codes of the form.

Appendix-K – *Results of calculation for stresses on the welds and bolts of the support used for holding a pressure vessel in the workshop*

Force on ii th bolt (F _i) :		
ii	Force	Stress
1	27767.07	2.73E+07
2	36279.43	3.56E+07
3	39268.57	3.86E+07
4	36279.43	3.56E+07
5	27767.07	2.73E+07
6	15027.43	1.48E+07
7	0.00	4.73E-09
8	-15027.43	-1.48E+07

ii	Force	Stress
9	-27767.07	-2.73E+07
10	-36279.43	-3.56E+07
11	-39268.57	-3.86E+07
12	-36279.43	-3.56E+07
13	-27767.07	-2.73E+07
Sum =		0

Stress on weld at position (θ) :		
$\varphi + \alpha =$	35	
$360 + (\varphi - \alpha) =$	325	
θ (degree) [($\varphi + \alpha$) -- 360 + ($\varphi - \alpha$)]	θ (radians)	Stress
35	0.61	9.37E+06
45	0.79	1.16E+07
55	0.96	1.34E+07
65	1.13	1.48E+07
75	1.31	1.58E+07
85	1.48	1.63E+07
95	1.66	1.63E+07
105	1.83	1.58E+07
115	2.01	1.48E+07
125	2.18	1.34E+07
135	2.36	1.16E+07
145	2.53	9.37E+06
155	2.71	6.91E+06
165	2.88	4.23E+06
175	3.05	1.42E+06
185	3.23	-1.42E+06
195	3.40	-4.23E+06
205	3.58	-6.91E+06
215	3.75	-9.37E+06
225	3.93	-1.16E+07
235	4.10	-1.34E+07
245	4.28	-1.48E+07
255	4.45	-1.58E+07
265	4.63	-1.63E+07
275	4.80	-1.63E+07
285	4.97	-1.58E+07
295	5.15	-1.48E+07
305	5.32	-1.34E+07
315	5.50	-1.16E+07
325	5.67	-9.37E+06
Sum =		6.33E-08

Average shear stresses:	
Avg. shear stress on bolt =	25921246.7
Avg. shear stress on weld =	6853273.33

Acknowledgements

I would like to thank Mr. Nilesh R. Raykar, my guide for the projects, for providing me with the opportunities of studying, learning and gaining practical experience in various fields during the period of training. His invaluable suggestions not only helped me to reach the successful completion of the tasks assigned, but also made me learn a lot.

I would like to give special thanks to Mr. Najeem Muhammad for helping me throughout with his wise suggestions, innovative ideas and whole-hearted help.

I want to thank Mr. Senthil Kumar, Mr. Yogesh Tandel, Mr. Rutvik Pawgi, Mr. R.B.Gosalkar, Mr. B.Viswanath and all others in the department who helped me during my work here.

And finally I would like to thank the HR department of Larsen & Toubro Limited, Powai Works and my institute, IIT Kharagpur, for giving me the opportunity to have a precious and rewarding experience of training in the prestigious industry of L&T Limited, Powai Works.

References

1. ASME Journal of Pressure Vessel Technology.
2. ASME codes for Pressure Vessel design.
3. *'Design by Analysis CD Manual'* issued by European Pressure Equipment Research Council, University of Strathclyde.
4. Elasto-Plastic Analysis document, Nitin Joseph, Technology Centre Process Plant & Technology.
5. Bickel and Ruiz, *'Pressure Vessel Design & Analysis'*
6. J. Spence and A.S.Tooth, *'Pressure Vessel Design Concepts and Principles'*