Design and Assessment of Low Cycle Fatigue Life of a Heavy Duty Press

A.V.Pradeep*, Kona Ram Prasad, CH.Jagadeesh*****

* (Department of Mechanical Engineering , S.V.P.Engineering College, Visakhapatnam)
**(Department of Mechanical Engineering , S.V.P.Engineering College, Visakhapatnam)
***(Department of Mechanical Engineering , S.V.P.Engineering College, Visakhapatnam)

ABSTRACT

Our project involves investigations on the failure of an 8000 ton heavy duty press. This press is used for forming boiler drums by U-shell pressing operation. The failure of the press had occurred due to formation of several cracks in the upper tool part of the press.

Initially the detailed three dimensional model of the upper tool is developed using the popular solid modeling package, Pro/E. Later meshing of the model is done in Hypermesh and analysis of the meshed model is carried out using the analysis package Ansys 8.0. The upper tool of the press has been analysed using

After getting the stresses and strain values in the upper tool body from the analysis, the expected low cycle fatigue life of the press is predicted using the Coffin-Manson's relations. These investigations have clearly brought out the cause of the failure of the press. The upper tool part of the press has a finite life design as regards cold correction operations, and, low cycle fatigue is the life consuming mechanism responsible for its failure.

Further recommendations are made for extension of life of upper tool part of press by way of stiffening. The scope of the work is further enhanced by finding the real stress distribution and strains existing in the upper tool by Neuber's approximation method. Thereafter the reliability of the method is discussed.

1. INTRODUCTION

In the last three-four decades, with the advent of computer based design packages of solid modeling and FEA, the field of mechanical design has undergone a sea change. These methods have equally contributed in the failure analysis and life estimation of structures. Though commonly used, the real test of all these techniques lies in their application to real life problem. This thesis presents a real life problem of failure analysis of an 8000 T heavy duty Press. Detailed investigations are carried out to analyze the failure of press. Also design modifications are suggested for its life extension.

1.1 OBJECTIVE OF THE PROJECT

The main objective of the project is to investigate the failure and suggest remedial measures for extending the life of the press.

1.2 WORK CARRIED OUT

The design of the upper tool part of the press has been analyzed using FEA for the extreme load conditions to which the press is subjected. The work comprises of preparing detailed three dimensional models using solid modeling packages like **Pro/E** and analysis of the existing design using analysis package like **Ansys** along with meshing packages like **Hypermesh**. The stress analysis involves elasto-plastic analysis. The complexity of the analysis required use of advanced concepts like elasto-plastic analysis, Sub modeling, use of macros etc. After the analysis, the expected life of the press is predicted.

These investigations have clearly brought out the cause of the failure of the press. The upper tool part of the press has a finite life design as regards cold correction operations, and, low cycle fatigue is the life consuming mechanism responsible for its failure. Recommendations are made for the extension of life of the upper tool part of press by way of stiffening.

The suggested modifications strengthen the upper tool and thereby reduce the stresses resulting in extension of life. This work involves many trials of stiffening arrangements and their analyses to work out the most suitable arrangement.

The scope of the work is further enhanced by carrying out basic studies on the application of the method of Sub modeling in elasto-plastic analysis. Also studies are made in respect of methods of computation of elasto-plastic strains using Neuber's approximation method.

2.ELASTO-PLASTIC ANALYSIS AND SUBMODELING

2.1 Elasto-Plastic (Nonlinear) Analysis:

The elasto-plastic analysis is the analysis which takes into consideration the deformations of the body unlike the elastic analysis, which assumes that the body is perfectly elastic. The elastic analysis is valid only in the situations where the stresses induced in the body are below the yield value of the material. There is always a permanent deformation when the yield value of the material is crossed.

In those situations performing the elasto-plastic analysis becomes very much necessary. It gives the exact values of the stresses and strains taking into consideration the deformations. One of the primary purposes of theory of plasticity is the explanation of the deformational behaviour of the materials stressed beyond the elastic limit. A second purpose is to determine the stresses at which the plastic behaviour initially begins.

The discussion of elasto-plastic analysis is started taking the classical example of the overhanging beam. Later it is extended to the press problem. The geometry of the beam along with the boundary conditions is as shown in figure 2.1

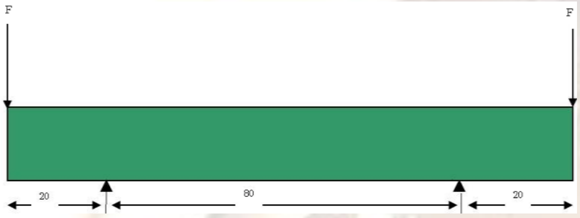


Fig – 2.1

The following attributes are specified in Ansys for the present problem;

1. Element Type: Solid 42,i.e. Plane element with two D.O.F
2. Material Properties:
   Elastic Modulus; E = 169000 N/sq.mm
   Poisson's Ratio = 0.3
   Yield Stress = 310 MPa
   Tangent Modulus = 8450

The value of tangent modulus is generally taken in between E/20 and E/30. Here it is taken as E/20.

Load Step	Force in Newton	Stress in X-direction (MPa)
1	200	243.589
2	250	304.486
3	300	328.922
4	350	316.681
5	200	192.488
6	0	78.155

The results for the six load steps are shown in the figures 2.2 to 2.8.

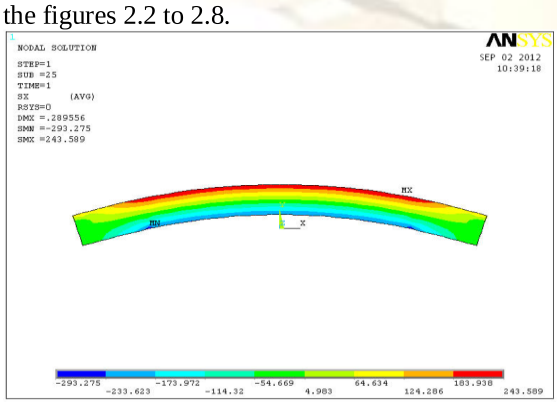


Fig – 2.2: At load 200 N

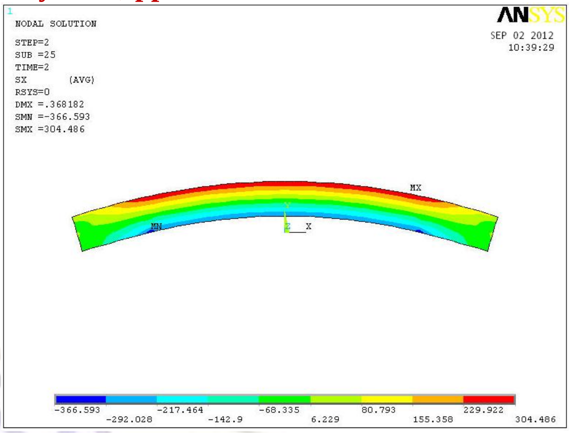


Fig – 2.3: At load 250N

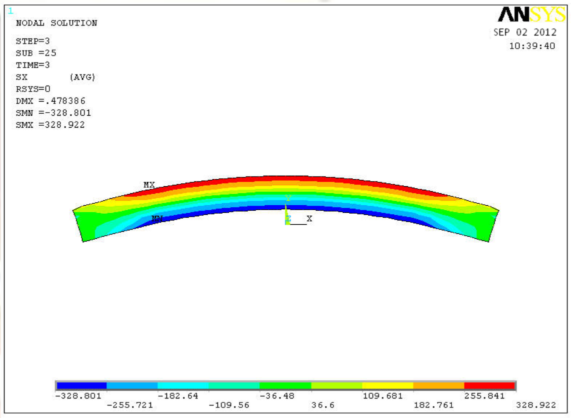


Fig – 2.4: At load 300N

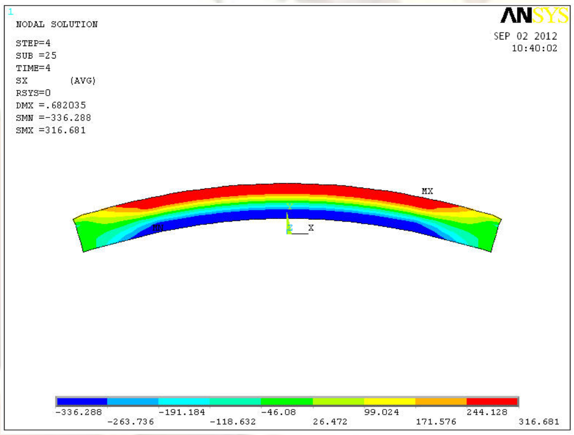


Fig – 2.5: At load 350N

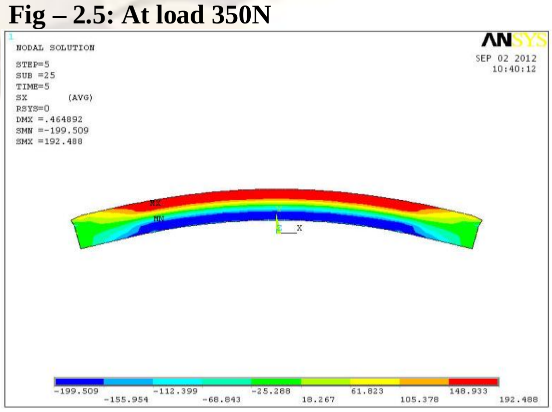


Fig – 2.6: At load 200N

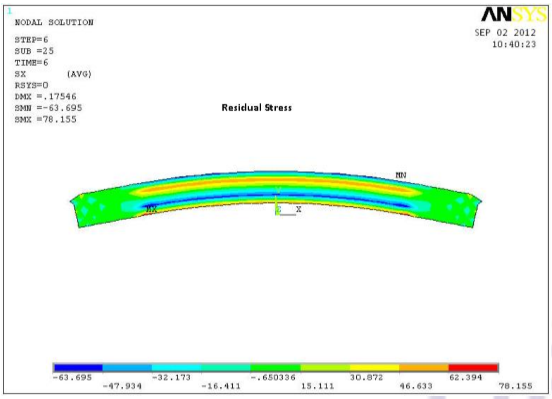
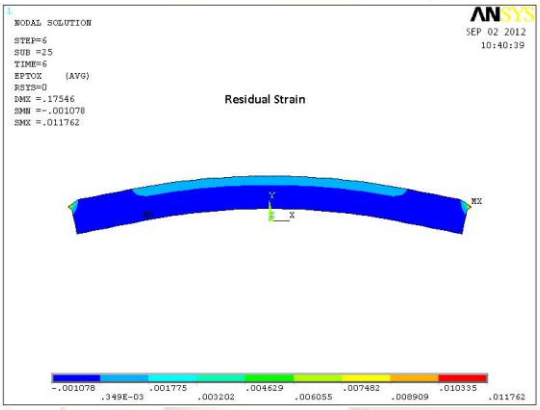


Fig – 2.7: At load 0N Residual stress



Fi

Fig – 2.8: At load 0N Residual strain

The yield stress of the material is given as 310 MPa. So in the third load step the yield value is crossed and the elastoplastic method is used by the software. Till the yield value is reached the elastic analysis only is performed.

Once the yield value is crossed the elastoplastic attributes are considered and the analysis proceeds as the elastoplastic analysis taking into consideration the permanent deformations.

It is also observed that the stresses for the first load step and the fifth load step are different even though the values of the forces are same i.e. 200N. This is because of the elastoplastic considerations. Initially there is no deformation for first load step. But for the fifth load step there is some deformation and due to this stresses are reduced. In the sixth load step even though there is zero loading there is some stress remaining. This is the residual stress in the beam.

2.2 Concept of Submodeling

Submodeling is a finite element technique used to get more accurate results in a region of the given model. Often in finite element analysis, the finite element mesh may be too coarse to produce satisfactory results in a region of interest, such as a stress concentration region in a stress analysis .The results away from this region, however, may be adequate.

To obtain more accurate results in such a region, you have two options:

(a) Reanalyze the entire model with greater mesh refinement, or

(b) Generate an independent, more finely meshed model of only the region of interest and analyze it. Obviously, option (a) can be time-consuming and costly (depending on the size of the overall model). Option (b) is the Submodeling technique

Submodeling is also known as the cut-boundary displacement method or the specified boundary displacement method. The cut boundary is the boundary of the submodel, which represents a cut through the coarse model. Displacements calculated on the cut boundary of the coarse model are specified as boundary conditions for the submodelInitially the concept of Submodeling is illustrated with the classical problem of a **Plate with a central hole** (Later it will be extended to the main problem of 8000T Press). The geometry is symmetrical about X andY-axis

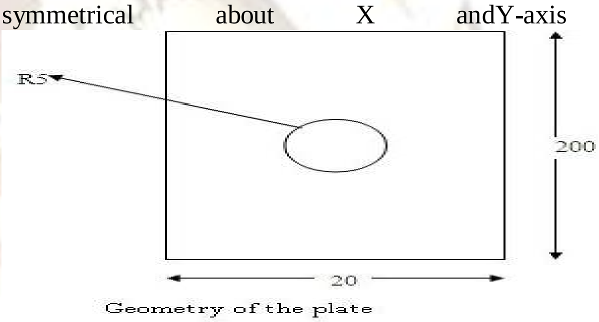


Fig – 2.9

For the purpose of analysis only the upper right quarter portion of the plate is considered which is as follows;

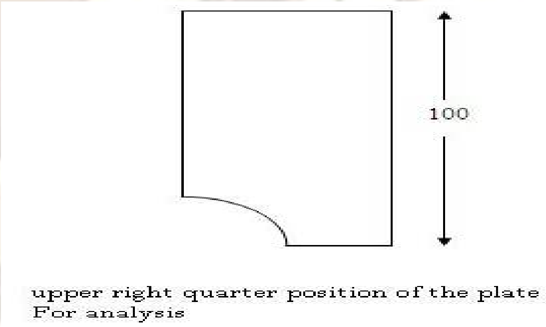


Fig – 2.10

2.3 Elasto-Plastic analysis and Submodeling for the plate with hole:

Initially the full geometry of the 2-D plate is created in Ansys 8.0, by creating keypoints, lines and areas. The geometry is then meshed with **coarse meshing**. The element type selected is Solid 42, i.e Plane element with two degrees of freedom. Meshed model is shown in figure 2.11;

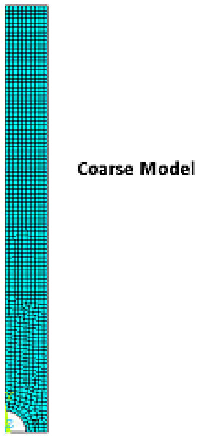


Fig – 2.11

The following material properties are specified for performing the Elasto-plastic analysis;

a. Elastic modulus = 169000 MPa
b. Poisson's Ratio = 0.3
c. Yield Stress = 310 MPa
d. Tangent Modulus = 8450

As the concept of Elasto-Plastic analysis is previously discussed the presentation directly proceeds with the present problem of concern.

Due to symmetry of the geometry only the quarter portion of the plate is considered for the analysis purpose. As the upper right quarter of the plate is considered, the following constraints should be applied on the plate;

a. Constraints of symmetry on the left vertical face
b. Constraints in Y-direction on the bottom face

The geometry, mesh and the displacement boundary conditions are ready for our analysis purpose. Now apply a **negative pressure** (i.e. tensile pressure) on the **top face** of the plate and see the stressdistribution.

Because it is Elasto-Plastic analysis the pressure is not applied in a single load step. Pressure is applied in five load steps with the following values;25MPa, 50 MPa, 75 MPa, 100 MPa, 125 MPa.

The Von-Misses stress distribution in the plate for all the five load steps are as follows;

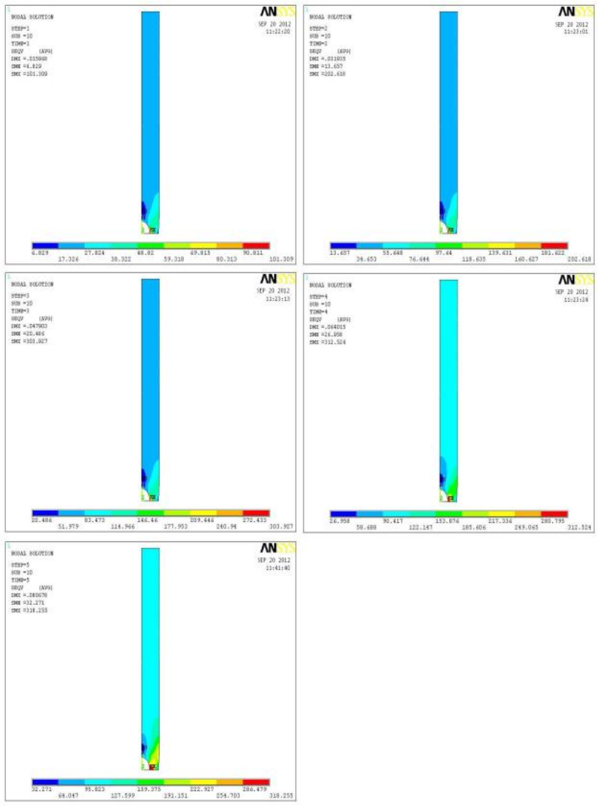


Fig – 2.12 to 2.16 Von-Misses stress distribution in the plate for the five load steps

CHAPTER 3
STRESS ANALYSIS OF UPPER TOOL OF THE PRESS

3.1 INTRODUCTION:

In this chapter the stress analysis concepts which were discussed in the preceding chapters, like Elasto-Plastic Analysis, Submodeling etc are applied to the upper tool of the press to find the actual stress distribution in the component.

3.2 SOLID MODEL OF THE UPPER TOOL:

The three dimensional solid model of the upper tool of the 8000 T press is modeled using the part modeling module of Pro/Engineer. Before modeling it must be ensured that the units are set to the required system. The units followed here are "mm" for length and "Newton" for force. The solid model of the central four limbs of the upper tool on which the total loading of 8000T is applied during cold working is shown in figure 3.1;

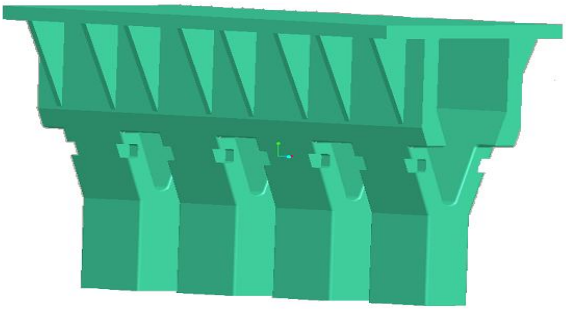


Fig – 3.1Solid Model Of The Upper Tool

It is evident from the above shown figure that there is symmetry in the geometry. So for the purpose of analyzing the upper tool all the four limbs need not be considered. The symmetry is exploited as illustrated in Fig. 3.2;

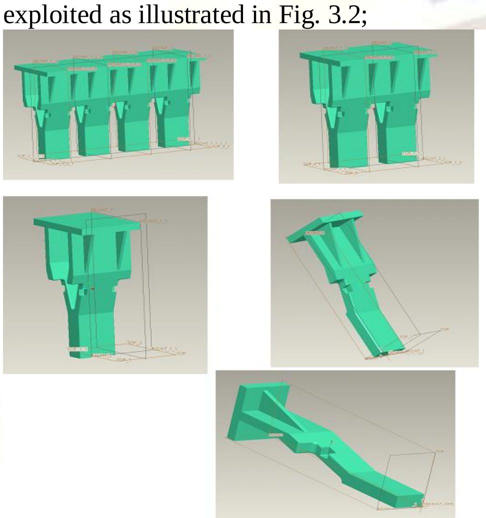


Fig – 3.2

Fig – 3.2 : The symmetry is exploited

The quarter model is considered for analysis purpose and here after only the quarter model is dealt with. The boundary conditions are selected based on the quarter part.

3.3 MESHING OF THE UPPER TOOL:

The geometry of the upper tool is first imported into Hypermesh from Pro/E. Hypermesh has the ability to read Pro/E files directly.

Also the “iges” file can be created in Pro/E and then read in Hypermesh. IGES stands for Initial Graphics Exchange Specification.

The geometry is then meshed with eight noded brick element by performing mapped meshing. The geometry along with mesh looks as shown in fig.3.3;

FIG -3.3 : Geometry along with mesh

After meshing is successfully done, the three dimensional brick elements are organised into a new collector and all other collectors containing the geometric entities and shell elements are deleted. The elements in the new collector are specified with an element type Solid 45, which is of first order type. Now the elements are exported to Ansys via data file.

3.4 IMPORTING THE MODEL INTO ANSYS AND PERFORMING ELASTIC ANALYSIS:

The ‘.dat’ file created in Hypermesh is imported into Ansys and the nodes and elements are available for analysis as shown in figure 3.4;

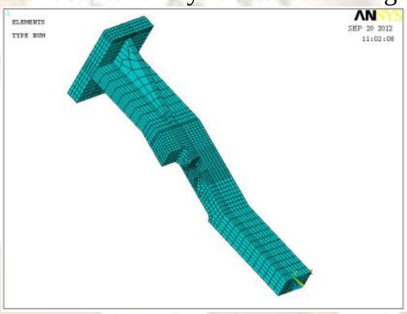


FIG - 3.4 : Nodes and elements are available for analysis

Now the boundary conditions and loads can be applied to the component. The following degrees of freedom constraints are applied on the F.E.model;

- Constraints in X-direction on the Z face nodes to account for the symmetry.
- Constraints in Y-direction on the top face nodes to fix the upper tool.
- Constraints in Z-direction on the X face nodes to account for the symmetry.

A pressure of **90.61 N/sq.mm** is applied on the bottom face of the tool. The pressure applied is 90.61 N/sq.mm because it is the equivalent value of pressure to the force of 500T acting on the quarter model.

The pressure is 90.61 N/sq.mm because;

Total load on four limbs = 8000T
Load per limb = 2000T
Load per half limb = 1000T
Load per quarter limb = 500T = 500000kg = 4905000 N

Equivalent Pressure = Load / Bottom area

= 4905000/54164.2

= **90.61 N/mm ²**

After the loads and boundary conditions are applied the F.E.model looks as follows;

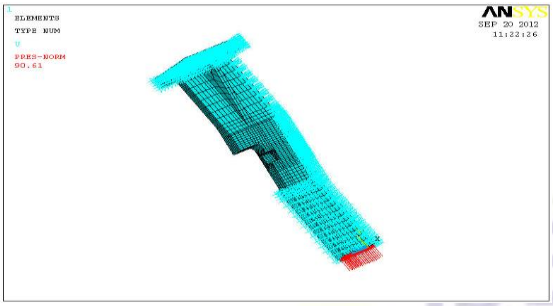


Fig – 3.5: F.E.model of the tool

The problem is completely defined and ready to be solved. So the problem is solved to obtain the solution. The results are shown below in fig. 3.6 and 3.7;

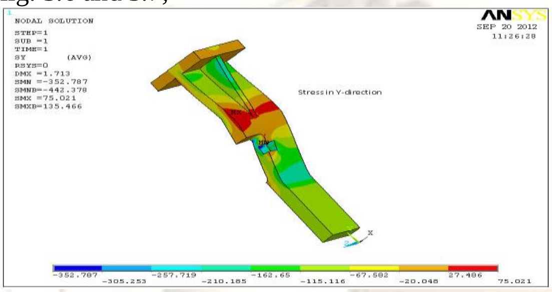


FIG – 3.6 : Stress in Y- Direction

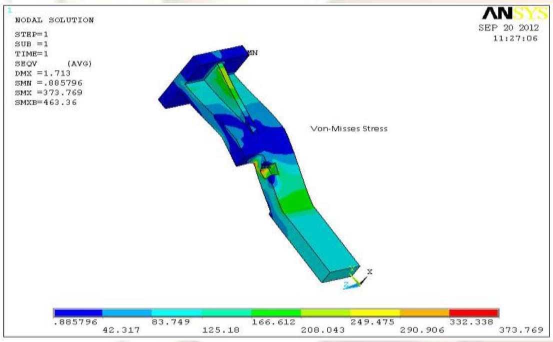


Fig –3.7: Von – misses stress

Performing a single analysis may not be sufficient to present the converged results in F.E.M. This can be said by looking at the percentage difference between the values of SMN and SMNB in the 'Stress in Y-direction plot' and SMX and SMXB in the 'Von-Misses Stress plot'. It is customary that the percentage difference is not allowed to be more than about 10%. But here the difference is around 20%. So it is needed to refine the mesh and resolve the problem. It is unnecessary to solve the problem with the total geometry. So the concept of Submodeling is followed to solve for the stresses at the critical regions. There are three regions on which concentration must be paid. They are shown in figure 3.8;

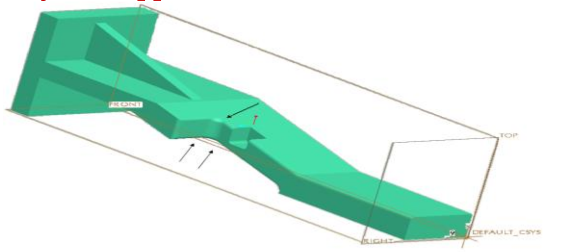


Fig – 3.8: Stresses at the critical regions on which concentration must be.

The submodel is selected and it looks as follows along with the mesh;

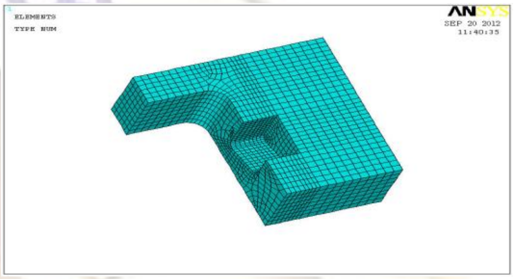


Fig – 3.9: Submodel of the selected region with the mesh.

Now the nodes on the top, bottom and right side face are selected and a node file is written. The window is closed and the previous problem with unconverged solution is opened. This is used as the coarse model for interpolating the boundary conditions. Here the node file is read and the load file is written. Again the submodel is opened and the load file is read. The interpolated boundary conditions are applied to the submodel in the form of degrees of freedom constraints. The submodel along with the boundary conditions is as shown in figure 8.10;

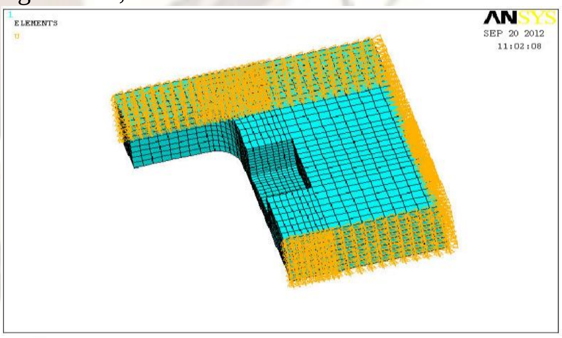


FIG – 3.10 : Submodel along with the boundary conditions

Now, giving the same material properties this problem is solved to obtain the results. The results are shown in the figures 8.11 and 8.12.

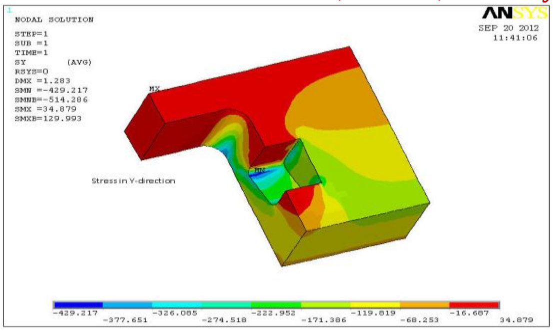


FIG – 3.11 : Stress in Y- Direction

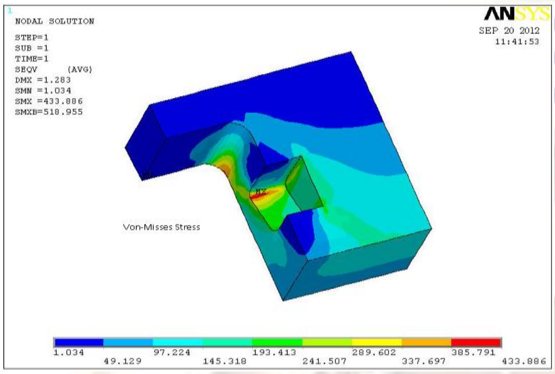


Fig – 3.12: Von – misses stress

Even this solution seems not converged because the percentage difference between SMN and SMNB in 'Stress in Y-direction' plot and SMX and SMXB in 'Von-Misses plot' is about 16%. So again the mesh is refined and the problem is resolved. The submodel along with new refined mesh and the results are shown in the following page. The same procedure followed previously is to be followed to obtain the solution.

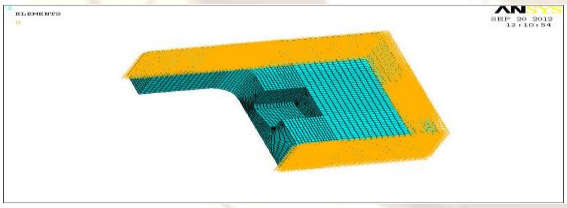


Fig – 3.13 Submodel along with new refined mesh

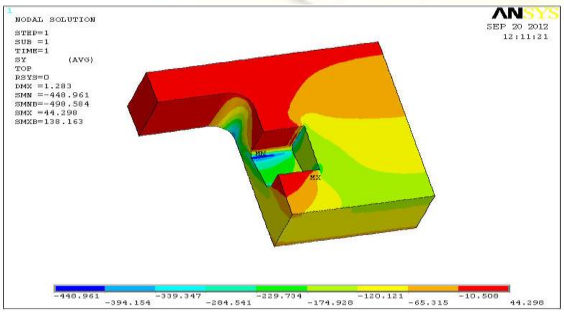


Fig – 3.14 : Stress in Y- Direction

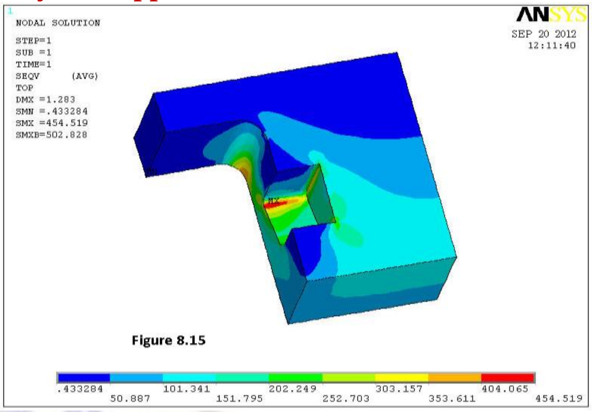


Fig – 3.15: Von – misses stress

It is clearly evident from the results (fig. 3.14 and 8.15) that the solution is converged. This is because the percentage difference between SMN & SMNB in 'Stress in Y-direction' plot and SMX & SMXB in 'Von-Misses Stress' plot is around 10%.

Also to cross check the convergence of the solution the submodel is again remeshed with finer elements and the problem is solved. It is observed that the results are more or less the same as the above results.

So the final results for the **elastic solution of upper tool** of the press are tabulated below:

Stress in Y-direction	Von-Misses Stress
- 448.961 MPa	454.519 MPa

3.5 ELASTOPLASTIC ANALYSIS PERFORMED ON THE UPPER TOOL OF THE 8000T PRESS

First the quarter portion of the upper tool is considered as done in elastic analysis previously and it is meshed with eight noded **coarse** brick elements in Hypermesh. Then the data file is created in hypermesh and that file is read into Ansys. So the elements and nodes are ready for analysis. Now the elastoplastic analysis is carried out on the upper tool with the necessary boundary conditions and the loads. Later the results are noted.

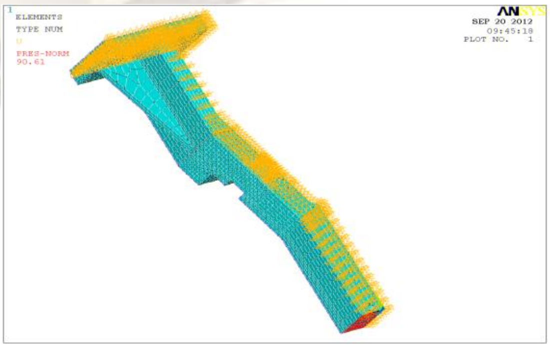


FIG - 3.16: The coarse model meshed in Hypermesh

1. Now a pressure of 50 MPa is applied at the bottom surface as the first load step.
2. Later three more load steps are applied as follows;
3. Now the four load steps are solved to obtain the results. The results are as follows;

As this meshing is **coarse** the solution cannot be considered as the converged solution. So it is tried to refine the mesh and solve the problem again and again till the converged solution is obtained. Unfortunately as the mesh is refined continuously the size of the problem goes on increasing tremendously to such an extent that the computer cannot solve the problem.

So the size of the problem needs to be reduced and at the same time a fine mesh is needed to get a converged solution. Now at this juncture the classic method of Submodeling is followed to reduce the size of the problem so that the computer can solve the problem without demanding for much memory space.

The problem is to be solved using both the concepts of Elasto-plastic analysis and Submodeling.

3.6 Elasto-Plastic Analysis and Submodeling Applied to the Upper Tool

As Submodeling is described and illustrated with an example previously, directly the list of the step by step approach for performing Submodeling along with Elasto-plastic analysis is provided.

Steps:

1. First it must be ensured that the Elasto-plastic solution for the full geometry (quarter section) with coarse meshing is available. This solution is any how ready in the previous section 8.5.

2. Suitable submodel must be selected so that all the critical regions are available in it.

3. Now the submodel is meshed with finer mesh (8 noded brick elements) than the full model using **Hypermesh** and later the nodes and elements are transferred into **Ansys.** Same mesh size is maintained as in the case of elastic Submodeling done previously.

4. At this stage the submodel is ready in Ansys and is as shown in figure 3.17;

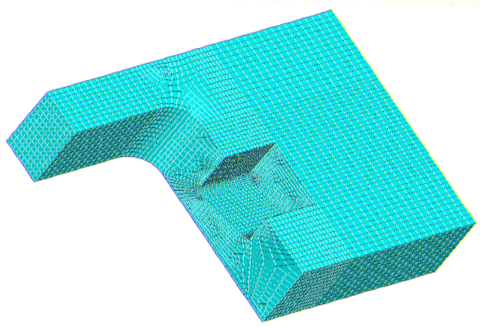


FIG – .3.17: Submodel in ansys

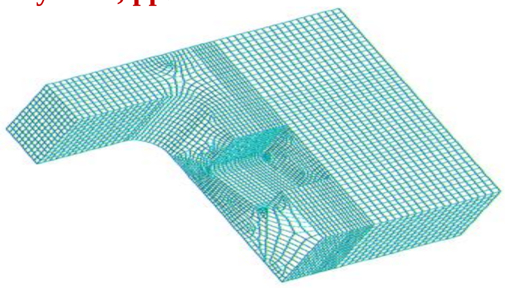


Fig - 3.18: Submodel in hyper mesh

5. Now select the nodes on the top, bottom and right face of the submodel as shown in figure 8.19;

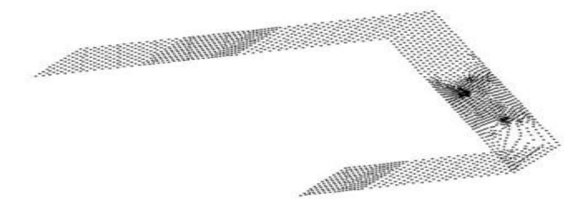


FIG – 3.19: Nodes of the submodel

6. Now write the nodes file with the following command in Ansys;

nwrite, submodel, nod

Here **'submodel'** is the name and **'nod'** is the extension for the node file.

7. Close the window and open the coarse full model, which is solved and has some preliminary solution.

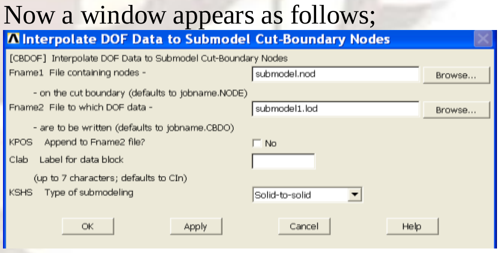
8. Here it must be ensured that the solution is in the first load step. To ensure this, read the results of the first load step through GUI (or) use the following command;

lsread, 1

9. Later the following procedure must be followed,

General Postprocessor > Submodeling > Interpolate D.O.F

Now a window appears as follows;



Here the node file is provided as input and the loads at the top and bottom surfaces of the submodel are interpolated from the full model. All the loads are interpolated as degree of freedom constraints and the load file for the first load step is written with the name **'submodel1.lod'**.

10. Thereafter steps 8 and 9 are repeated for all the remaining load steps and the following load files are written;

submodel2.lod,submodel3.lod, submodel4.lod

11. Now the full model with coarse mesh is closed and again the submodel is opened. Suitable material properties are provided and then the following steps are followed,
The following command is given;

***use, submodel1.lod**

Then the loads interpolated from the first load step in the full model are applied in the form of degrees of freedom constraints on the submodel. Now write this loading situation as the first load step.

Now read the remaining load files as follows subsequently writing the second, third and fourth load steps after each load file is read;

12. Now solve all the four load steps to get the solution. Here the submodel with the loading situation interpolated from the full coarse model is being solved.
13. The results obtained with the submodel will tally with those results obtained if the full model is solved with the same mesh size as that of the submodel. This was experimentally showed previously taking the example of plate with hole.
14. The results of the submodel i.e. the upper tool of the press are as follows;

Load Step No:	Pressure Applied	Stress in Y-direction	Von-Misses Stress
1	50	-247.743	250.811
2	75	-325.887	318.323
3	**90.61**	**-347.741**	**319.541**
4	125	-378.494	337.558

With a pressure of 90.61 N/sq.mm the stresses are as shown above in bold. Also the values of stresses and total strain are tabulated below;

Pressure Applied	Stress in Y-direction	Von Misses Stress	Total Strain
90.61 MPa	-347.741 MPa	319.541 MPa	0.002851

3.7 NEUBER'S APPROXIMATION METHOD TO FIND ELASTO PLASTIC SOLUTION:

Neuber's rule is used to convert an elastically computed stress or stain into the real stress or strain when plastic deformation occurs. The Neuber's method of finding the elasto-plastic solution is a formula based mathematical approach by which one can find the elasto-plastic solution without performing the elasto-plastic analysis.

This is an approximation technique that can be used to take into account the redistribution of stress caused by plastic flow in a zone of stress concentration, a case often met in practice. The conditions of validity of this method are that the plastic zone must remain sufficiently contained (surrounded by elastic zone) and that the loading must be radial. The elastic solution of the problem under the action of the external forces is assumed to be known. The method can be conveniently explained with the help of the following figure 8.20;

The elasto-plastic solution can be found out from the Neuber's curve by looking at the point where it cuts the Stress-Strain curve. The Neuber's hyperbola must be constructed such that it travels from the elastic solution curve to the stress-strain (hardening) curve.

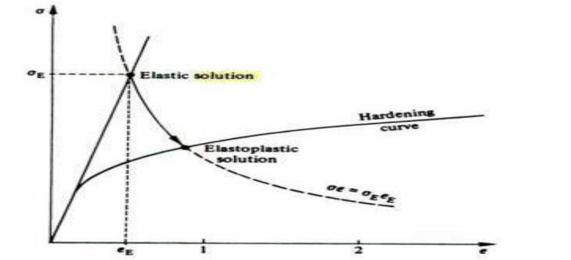
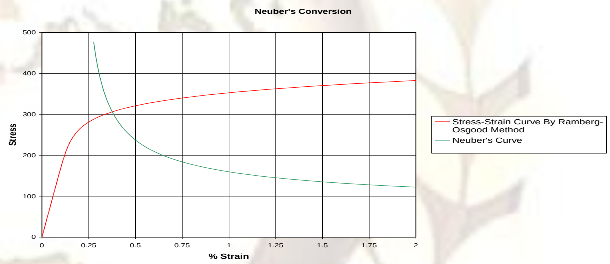


Fig – 3.20 Stress-strain (hardening) curve

After getting the Strain and Stress values they are plotted with suitable scale on the same graph on which the Ramberg-Osgood stress-strain curve is plotted. The point where both the curves meet gives the elasto-plastic solution. In our present case;



From the above graph in figure 3.21 the following results can be drawn;

Equivalent Total Strain	0.35	%
Elastic Strain	0.18	%
Plastic Strain	0.17	%
Real (elasto-plastic) Stress	320	MPa

These results are nearer to the results obtained in the Elasto-Plastic analysis. The results do not match exactly because this is an approximation method and also does not take into consideration the geometry and the loading conditions of the problem. The method evaluates the solution based on the value of the elastic stress.It is evident from the five stress distributions that the maximum stress is always prevailing at the same point due to stress concentration. The values of the maximum stress for the five load steps are tabulated below;

Coarse Mesh Results

Load Step Number	Pressure Applied On the Top Face (MPa)	Von-Misses Stress (MPa)
1	25	101.309
2	50	202.618
3	75	303.927
4	100	312.524
5	125	318.255

properties are assigned same as the coarse model to the submodel also.

Let the file name for the submodel be assigned as **submodel.db**

The submodel's geometry is created in Ansys 8.0 and it is meshed with fine elements. The geometry along with the mesh is shown in the figure 3.22.

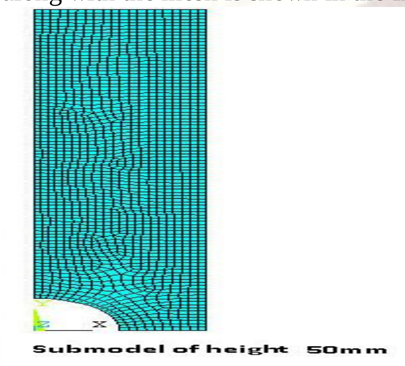


Fig – 3.22

Now, from this submodel the nodes on the top face are selected and written to a node file with the following name and extension;

In the second version of design enhancement, the main objective is to reduce the thickness of the plate welded to the body of the upper tool, in the successful model of the first version of design because the plate thickness is 50mm and welding such a big plate with 50mm thickness practically poses many problems during and even after welding.

CHAPTER 4
MODIFICATION IN DESIGN OF THE UPPER TOOL

4.1 Introduction:

Before discussing about the modifications done to the existing design a glance at the existing model of the press is required. The stress values observed in the analysis are also tabulated below;

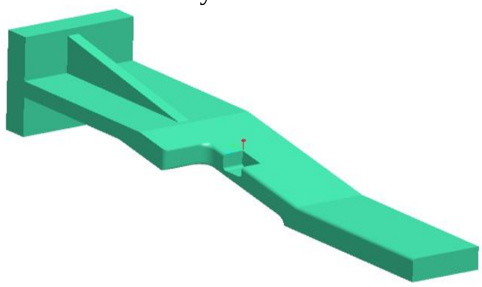


Fig – 4.1: First normal solid model

Max Tensile Stress(MPa)	Max Compressive Stress(MPa)	Von-Misses Stress(MPa)
44.298	- 448.961	454.519

Now the following modifications are made to the design to reduce the max compressive stress value to less than 250MPa. The stress value is tried to be kept below 250MPa taking into consideration the low cycle fatigue, to which the component is subjected to.

4.2 FIRST VERSION OF DESIGN ENHANCEMENT

1. Half Part with Middle Cut Rib:

In this model one rib of 50mm thickness and 150mm height is provided in the half symmetry portion and the other rib remains the same as in the original model. The middle rib is cut at the top portion to provide provision for holes. The figure and the results are shown;

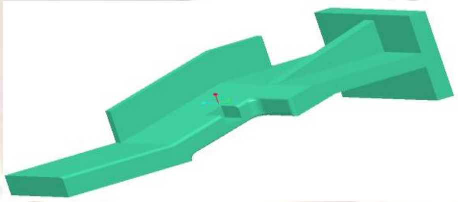


Fig –4.2: Half part with middle cut rib

Max Tensile Stress(MPa)	Max Compressive Stress(MPa)	Von-Misses Stress(MPa)
85.678	-381.377	359.449

2. Part with Back Plate:

In this model a back plate of 25mm thickness is provided to the body. The stresses are as shown in the table;

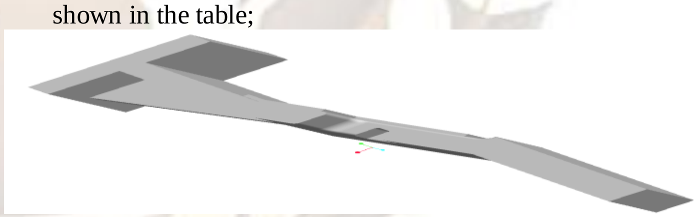


Fig – 4.3 Part with back plate

3. Outer Plate With Middle Rib:

This model consists of an outer plate of 50mm thickness and a center rib of 50mm thickness (25mm due to quarter symmetry) and 150mm height. The stresses are as tabulated below;

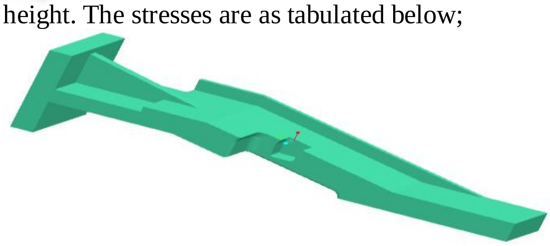


Fig – 4.4: Outer Plate With Middle Rib

CHAPTER 5
DISCUSSION OF RESULTS

After the accomplishment of the detailed stress analysis and life calculations the following results were arrived at:

➢ The maximum stress values in the upper tool of the press observed in the Elastic Analysis along with Submodeling are as follows;

STRESS IN Y-DIRECTION	VON-MISSES STRESS
- 448.961 MPa	454.519 MPa

➢ The above values are far more than the yield value of the upper tool material which is 310MPa and hence the Elasto-plastic analysis is carried out along with Submodeling to find the actual values of stresses and strains.

The actual stress values observed in the Elasto-Plastic Analysis are as follows;

➢ Now the Elasto-plastic solution is found out using the Neuber's method. As mentioned earlier this method aims at finding the Elasto-plastic solution without actually performing it, rather the solution is arrived at based on empirical relations taking the elastic stress as the input. The Elasto-plastic solution found out by this method produces the following results;

Neuber's Elasto-plastic stress	Neuber's Elasto-plastic strain
320 MPa	**0.35**

By comparing the above results with the results obtained in Elasto-plastic analysis it can be said that the Neuber's method gave a satisfactory result. But it does not give exact results because the method does not take into consideration the geometry and the loading conditions of the problem. The only thing it considers is the value of the Elastic Stress. Also, this method is applicable only if the plastic stress is contained between elastic stresses.

➢ Based on the values of stresses and strains in the Elasto-plastic analysis the life of the press is estimated to be 22000 cycles using the Coffin-Manson's relations.

➢ By seeing the values of the stresses one can say that the existing design needs to be strengthened by providing modifications to the design. The following modifications are suggested to be provided in the upper tool as shown in Fig.--

a. A plate of 50mm is welded to the body of the upper tool.
b. Two ribs of 50mm thickness are provided on the plate.

The suggested modifications bring down the stress values considerably, which help in extending the life of tool. The stress values in the modified design of upper tool are as follows.

The stress values are observed to be satisfactory but the design is not practically appreciable because of the plate thickness being 50mm. It is not a good practice to weld structures of such high thickness and thus weight. This makes the welding weak.

Therefore the design is modified once again in the second version as follows;

a. A plate of 32mm is welded to the body of the upper tool.
b. Two ribs of 50mm thickness are provided on the plate.
c. A rib of 25mm thickness is provided in between the two ribs of 50mm thickness.

The stress values in the second version of design enhancement are as follows;

CHAPTER 6
CONCLUSIONS

After successfully completing the project the following conclusions can be made in accordance to the work done;

Rigorous Elasto-Plastic Analysis is carried out on the heavy duty press to predict the life. Submodeling is equally valid for Elasto-Plastic analysis as it is for the elastic problems. This is proved in the dissertation by taking the example of the plate with hole.

The Stress-Strain curve developed using the Ramberg-Osgood relations satisfactorily matches with the original Stress-Strain curve of the material. Neuber's Conversion Method to calculate the stresses and strains for elasto-plastic problems gives reasonably good results.

The Coffin-Mansion's relations provide a method satisfactorily to predict the life of engineering components. The finite life of the press has been consumed in twenty years due to the excessive operations involving high loads.

Modification is done to the design of the upper tool to ensure that it can withstand the loading conditions. The modified design can satisfactorily withstand the loading conditions.

CHAPTER 7
RECOMMENDATIONS FOR FUTURE WORK

No work in the universe has a final solution or conclusion. So it will be hypocritical to say, the project work is complete. There is always refinement to solutions in any calculations, especially in engineering calculations. There is lot of scope for work to be carried out in the case of the press even though lot of work has been done. A lot of research work can be done in the following areas;

1. Even though the optimum geometry is proposed the Optimization of the geometry can be carried out using the available Finite Element Packages.
2. The Optimization of the weight can also be carried out using the Finite Element packages.

3. The Life Calculation is done in the present work using the Coffin-Manson relations. This can be cross checked by performing the Life Calculation using the popular Finite Element Packages. Also the life can be calculated for the modified design.

REFERENCES

1. Low Cycle Fatigue and Life Prediction: Amzallag, Leis and Rabbe Publisher A.S.T.M International.
2. Journal of Applied Mechanics and Technical Physics C/C of Zhurnal Prikladnoi Mekhaniki I Tekhnicheskoi Fiziki 28
3. Journal of Strain Analysis For Engineering Design
4. Transactions - American Society of Mechanical Engineers Journal of Engineering Materials and Technology
5. Materials Science and Engineering (Lausanne)
6. Transactions - American Society of Mechanical Engineers Journal of Engineering Materials and Technology
7. International Journal of Pressure Vessels and Piping
8. Fatigue and Fracture of Engineering Materials and Structures