University of Strathclyde Glasgow

Strathprints Institutional Repository

Development and Implementation of the Abaqus Subroutines and Plug-in for Routine Structural Integrity Assessment using the Linear Matching Method.

James Ure[(A)], Haofeng Chen[(A)], David Tipping[(B)]

[(A)]Dept of Mechanical and Aerospace Engineering, University of Strathclyde, Glasgow, G1 1XJ

[(B)]Central Engineering Support, Existing Nuclear, EDF Energy, Barnwood, Gloucester, GL4 3RS

Abstract: In recent years the Linear Matching Method (LMM) has been developed as a tool for structural integrity assessments of components subjected to cyclic loading conditions. Its capabilities include, among others, calculation of the shakedown limit, ratchet limit, plastic strain range for low cycle fatigue, creep rupture time and fatigue creep interaction. The LMM is now incorporated into EDF Energy's R5 research program for the high temperature assessment of structural components.

The purpose of this paper is to describe the development of the LMM framework, its incorporation into Abaqus and current plans to take the method from being primarily research based into wider use by industry for routine structural assessments.

The LMM calculations are primarily carried out using the UMAT subroutine, and the first topic discussed in this paper is the implementation of this user subroutine. This includes details of the coding scheme to allow use of multi-processors for the calculations. A brief comparison of the LMM with full cyclic FEA is also included to validate the method and to demonstrate its advantages. The second topic of this paper discusses the development of an Abaqus/CAE plug-in to aid wider adoption of the LMM as an analysis tool for industry. The structure of the plug-in is described alongside the processes used for data collection from the user and automatic configuration of the model.

Keywords: Shakedown, Ratcheting, Plug-in, UMAT Subroutine

1. Introduction

Plant components will very commonly operate under varying loading conditions, where transients of pressure and temperature during both startup/shutdown and normal operating conditions can induce large cyclic stresses in the structure. When analysing the integrity of these components it is often necessary to determine the steady state response to the cyclic loading so that appropriate subsequent analyses and checks can be performed. Figure 1 shows the possible responses to applied cyclic loading on an interaction diagram, also known as a Bree diagram (Bree, 1967), where the loading is decomposed into steady state (X-axis) and cyclic (Y-axis) components.

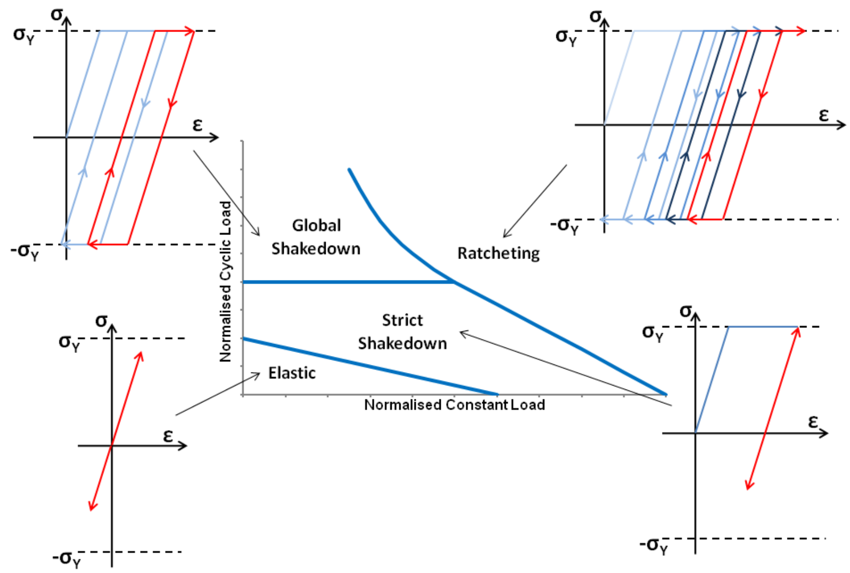

Figure 1. Possible Responses to Cyclic Loading

When the level of applied loading is small enough a wholly elastic response is observed everywhere in the structure. In the strict shakedown region (also known as elastic shakedown) the applied loading causes yielding and plastic strains in the first few cycles. The initial yielding causes a residual stress field to form which then prevents any further plastic strains developing, resulting in an entirely elastic response during subsequent load cycles. When a structure is in global shakedown (also known as plastic shakedown) some yielding occurs in every cycle. The nature of this yielding in some areas of the structure is such that the plastic strains form a closed cycle i.e. no accumulation of plastic strain from one cycle to the next. Ratcheting is where the application of the loads during the cycle causes yielding in every cycle, but the plastic strains do not form a closed loop, and therefore will continue to increase leading to eventual failure of the structure. However, if the initial application of the loads is large enough then the structure may reach its plastic collapse limit state. In structural integrity assessments, showing that a ratcheting response and plastic collapse are not going to occur is very important.

In the UK nuclear industry the R5 assessment procedure (Ainsworth, 2003) is used to perform high temperature structural integrity assessments of structures with creep and fatigue loading. The majority of R5 is based on simplified methods and uses elastic analyses as the basis of the procedure. If this route is found to be overly pessimistic then cyclic inelastic Finite Element Analysis (FEA) is necessary. The drawbacks to inelastic FEA are the difficulty in obtaining accurate material data (both time independent plasticity and time dependent creep models) and the ambiguity often present when deciding if a stable cycle has formed or not. A solution of many cycles may be required to determine the shakedown status of the component, which has obvious time and computational costs.

Several new methods, collectively named "Direct Methods" have been developed to produce increasingly accurate shakedown limit boundaries. One of the most successful direct methods is the Linear Matching Method (LMM) (Chen, Ponter and Ainsworth, 2006a; 2006b) which is implemented in Abaqus through user subroutines. The LMM offers several advantages over cyclic FEA including the ability to provide a definitive shakedown/non-shakedown solution, significant improvements in computational effort and the convenience of using of the same material data as used in the elastic route in R5.

EDF Energy have incorporated the LMM into their R5 research programme with a view to including it as an alternative to the simplified and full FEA methods. At present however the method remains primarily a research tool. Barriers to wider adoption stem from having to learn how to effectively use the subroutines and the need to carry out code changes for each analysis. In order to resolve these problems, work is currently under way to create a user interface which will gather the necessary information from the user before automatically configuring the model and subroutines for the LMM analysis. This user interface will be implemented using the Abaqus plug-in framework.

The purpose of this paper is to give an overview of the Linear Matching Method and its development as a structural integrity tool. A brief theoretical introduction is followed by an example of the accuracy of the method when applied to a realistic engineering structure. The structure of the Fortran subroutines is then explored in more depth, and finally an overview is given of the implementation of the user interface plug-in.

2. LMM Theory and Verification

2.1 Theoretical Background

The premise of the LMM is that a nonlinear material response can be replicated by a series of linear elastic calculations where the elastic modulus is modified over the volume of the structure (Chen and Ponter, 2001). A linear elastic calculation is performed for each load extreme in the load cycle. Where the stresses are greater than the yield stress the modulus is reduced. This process is demonstrated in Figure 2a. The reduction in modulus is such that the effective stress at that point is reduced to the yield stress (keeping the total strain level the same). The next increment then uses the updated modulus value from the previous increment which allows the stresses to redistribute in the structure as shown in Figure 2b and c.

The LMM uses the shakedown bounding theorems (Koiter, 1960; Melan, 1936) to determine the shakedown status of the model. The upper bound theorem of Koiter is the primary model used by the LMM, which uses an energy balance between the external work from the applied loads and the internal energy dissipation. At the end of each increment Koiter's theorem is applied and a load multiplier is calculated. This multiplier is used to scale the applied loads in the next increment. The combination of the modulus adjustment and the scaling of the loads produce a series of solutions that converge towards the exact shakedown limit. Convergence of the procedure is judged by the difference between consecutive load multipliers.

Apart from the shakedown limit, at the converged state, the LMM calculations provide a wealth of information to the user according to the analysis type, including the residual stress field, the varying residual stress field and plastic strain range (for a global shakedown analysis), creep modified shakedown, creep rupture and creep fatigue interaction (Chen and Ponter, 2010; Chen, Chen and Ure, 2012).

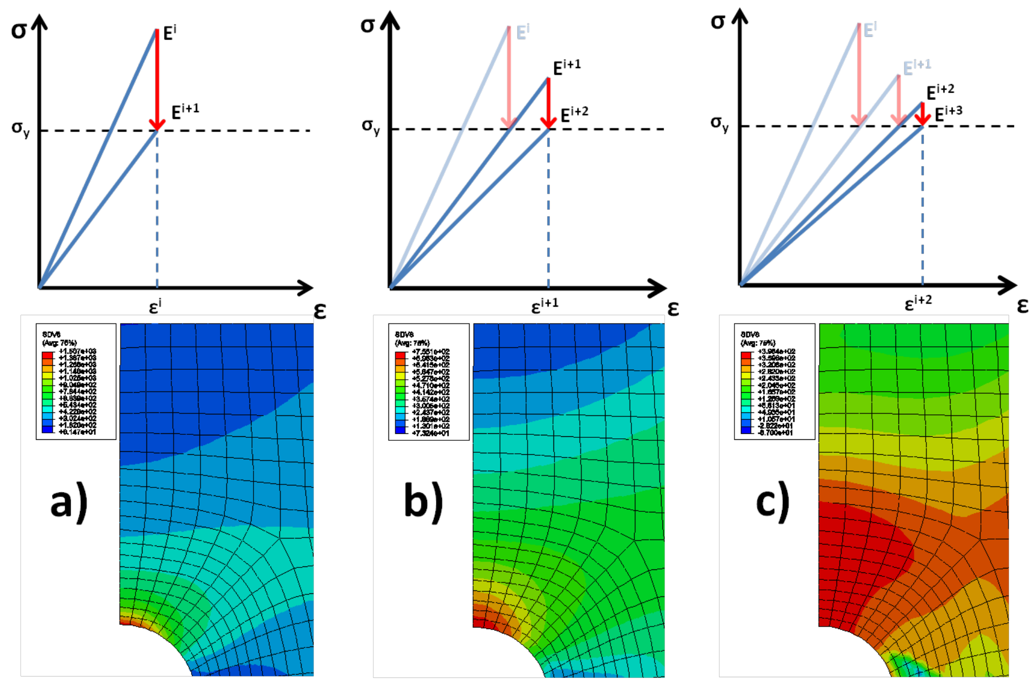

Figure 2. Modulus Adjustment Procedure and Resulting Stress Redistribution

2.2 Verification of the LMM

Many studies have been undertaken to verify the accuracy of the LMM. An example from (Ure, Chen and Tipping, 2012) of a pipe intersection quarter geometry and results are shown in Figure 3. The intersection, made from three materials as shown in Figure 3a, was subjected to steady state internal pressure and cyclic thermal loading (which induced high stresses at the material boundaries due to differential thermal expansions of the materials).

The interaction diagram calculated by the LMM is shown in Figure 3b and shows the areas of strict shakedown, global shakedown and ratcheting calculated by the LMM. Four points (namely A, B, C and D in Figure 3b) were selected for verification by step by step analysis and the plastic strain histories of these analyses are shown in Figure 3c. Theses histories show that the points outside of the ratchet boundary (i.e. points B and D) show a clear ratcheting response. Point C in the global shakedown region shows a reverse plasticity response, and point A shows a strict shakedown response where the plastic strains stop accumulating after the first few cycles. In the case of point C the plastic strains are still accumulating after the 20 cycles shown in figure 3c. In fact the solution of over 1000 cycles was required to prove that a reverse plasticity response was achieved. Had the LMM boundary not been present point C may have otherwise been falsely interpreted as a ratcheting response.

Where the loading caused the intersection to be in global shakedown, further step by step analyses were performed to compare the plastic strain ranges with the LMM. The LMM results compared favourably with step by step, with never more than 1% difference in plastic strain range between both methods.

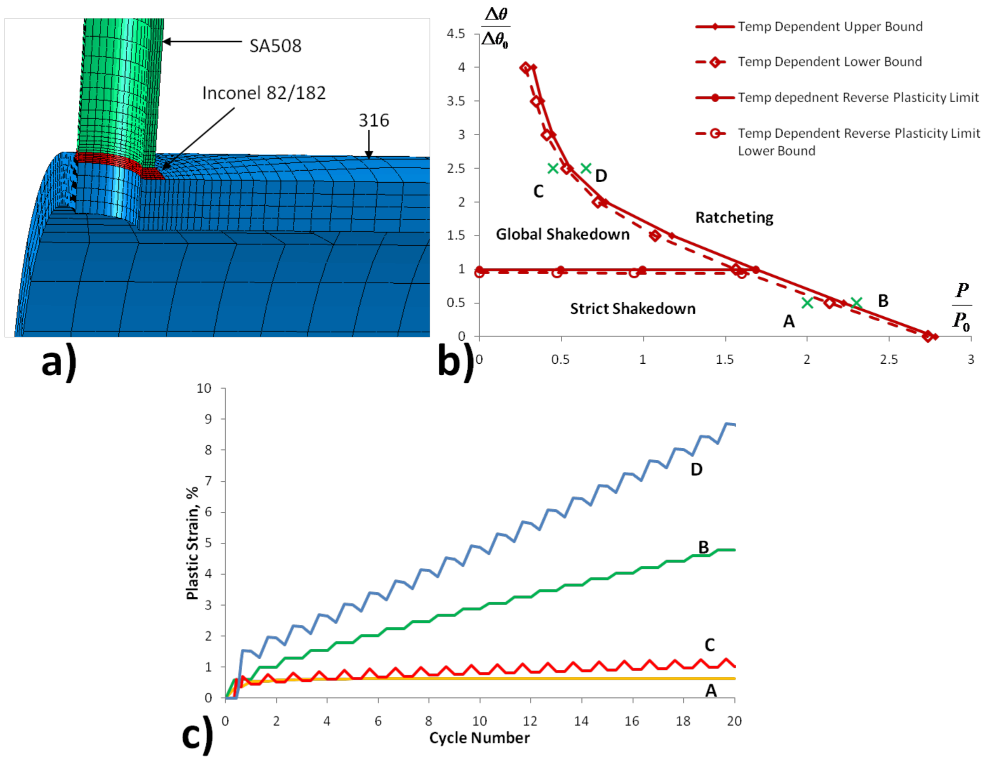

Figure 3. Sample Results from Analysis of a Pipe Intersection

In addition to the ability to evaluate the proximity of the predicted ratchet and shakedown boundaries, the LMM also has advantages in terms of solution time. The ambiguous results possible with step by step mean that many cycles must be solved to obtain results which clearly show the stabilised response of the structure. In the case of point C in the pipe intersection example the step by step analysis took around six times more CPU time than the equivalent LMM analysis to find this stabilised response.

3. LMM User Subroutines

The calculations and results described in section 2 are implemented in Abaqus through user subroutines. Several of the Abaqus user subroutines are used to perform the calculations and peripheral tasks, and an overview of the structure of these subroutines is given in this section. A flow diagram is given in Figure A1 of the Appendix which represents the interaction between the major subroutines pictorially.

3.1 Analysis Configuration

The LMM subroutines have been written in as flexible a manner as possible. Numerous options are possible, for example analysis of many different continuum element types (2D plane stress, plane strain, and 3D) and the initial configuration of the set of the subroutines to perform the required LMM analysis for the current model is performed in the UEXTERNALDB subroutine.

The UMAT routine has been coded for multi-core processing (see section 3.4) and this makes the configuration of other subroutines from within UMAT, whilst the calculation procedure is under way, very difficult. However, the UEXTERNALDB, which is called before the UMAT calculation process begins, is perfect for performing this initial configuration.

Within UEXTERNALDB the tasks performed include reading the model and the information input by the user from the plug-in (see section 4) to configure all subsequent subroutines to perform the correct analysis. For example, UEXTERNALDB uses the the load cycle information from the user to populate arrays of load multipliers which are then used within UMAT. Other properties of the model are detected automatically (the element type for example) and this is also used in subsequent subroutines (e.g. populating the jacobian). Once this configuration process is complete, UMAT and URDFIL can then carry out the calculations with no further input from UEXTERNALDB.

3.2 UMAT Routine

After UEXTERNALDB, the UMAT routine is called. Initially the UMAT is used to perform an elastic analysis for each of the applied loads and predefined fields in turn. The elastic stress components, effective elastic stress and the temperature at each integration point are stored for later use in the LMM calculation. After all elastic calculations have been performed a null step is carried out which has no function in the analysis other than to return all arrays to zero and act as a 'reset' before the LMM calculation begins.

The LMM calculation itself is an iterative procedure, and is carried out as multiple increments within a single Abaqus static general step. At the start of each iteration superposition of the previously calculated elastic stresses is performed to create the load cycle according to the information given by the user. At this stage the load cycle is also scaled by the load multiplier (calculated in the URDFIL routine during the previous iteration) as described in section 2.1. This gives the total elastic stress at each point in the load cycle.

With the load cycle constructed and scaled accordingly, the remainder of the LMM calculation can be carried out, which involves the modulus adjustment procedure shown in Figure 2 and calculation of the residual stress field. The important values and parameters are stored as state dependent variables (some are required in the next iteration, and others purely to create contour plots from the odb file). The LMM calculation also requires the energy values returned for access in URDFIL. As mentioned in section 2.1, many options exist at this stage of the analysis including calculation of creep rupture, creep modified shakedown and calculation of the plastic strain range if a global shakedown analysis is performed.

3.3 URDFIL Routine

When the LMM iteration has been carried out, the URDFIL is then called. The energy values calculated in the LMM calculation are retrieved as integrals over the volume of the structure from the .fil file. These energy values are used for two purposes. The first is to judge the convergence of the calculation and bring the procedure to a close if the convergence tolerance is met. The second use is to calculate the load multiplier for the next increment. If convergence has not been achieved, the procedure returns to the LMM calculation section of the UMAT to carry out a further iteration, and this is repeated until the convergence tolerance is met or the maximum number of increments is reached.

3.4 Implementation Considerations for use with Multiple Processing

With even the most basic of new desktop computers now equipped with multiple CPUs, the decision was made to make use of this by programming the UMAT subroutine for multiple core processing, specifically the Message Passing Interface (MPI) form. In order to achieve this, several limitations must be imposed on the code such as omission of SAVE and DATA statements.

The structure of the subroutines described in this section (and shown in the Appendix) is in some part a result of the choice to use multiple processing. In the original LMM subroutines the initial elastic analyses were conducted as a standalone analysis where the elastic stresses were written to text files. The LMM analysis subroutine would then read these text files and populate 'model sized' arrays with these stresses. With multiple cores now being used, reading data from text files in UMAT may become unpredictable and would almost certainly slow the analysis down, defeating the purpose of using multiple cores. The new structure of the subroutines sees the elastic analyses conducted in the same Abaqus job as the LMM analysis. The elastic stresses are now passed into the LMM analysis by storing them as state dependent variables in the odb file (STATEV array in UMAT). This tactic is also adopted when variables need to be saved for use in the next iteration (values such as strains and modulus for example). Storing information and passing it between iterations in this way removes the need for the SAVE and DATA statements and also means that the large arrays that would have stored this data for the entire model are not needed. Removal of these large arrays allows the code to execute faster even when running on a single CPU, giving additional solution time gains.

4. Plug-in Development

With the validity of the results and the analysis options in place, the major barrier to wider adoption in industry is the user-friendliness of the analysis procedure. For an engineer unfamiliar with the LMM procedure making the necessary changes to the subroutines for each new analysis can be a daunting task.

This situation was remedied to an extent by David Tipping (2008), who originally rationalised the LMM subroutines into a single procedure, allowing much of the required data to be input using a simple text file rather than changes to the code itself. The current project aims to take this process one stage further by creating a Graphical User Interface (GUI) via an Abaqus plug-in which will allow the user to input all analysis information in an intuitive way within the familiar Abaqus/CAE environment. This plug-in will take the information from the user and use it to automatically configure both the subroutines and the model itself so that a LMM analysis can be performed. This automation removes many of the sources of error possible in previous incarnations of the method, whilst at the same time providing a more convenient and faster way of using the LMM.

In terms of the model in CAE, the plug-in carries out modifications to format it correctly to match the subroutines. Upon first invoking the plug-in, several scripts query the model to determine if any problems would arise with the LMM process. This includes checks to warn against an invalid element choice, and to identify if there are any existing LMM modifications (as these will be overwritten).

If the model passes these checks then the user is presented with a series of dialog boxes to input data and select analysis options. The dialog boxes used within the current prototype of the plug-in are shown in Figure 4.

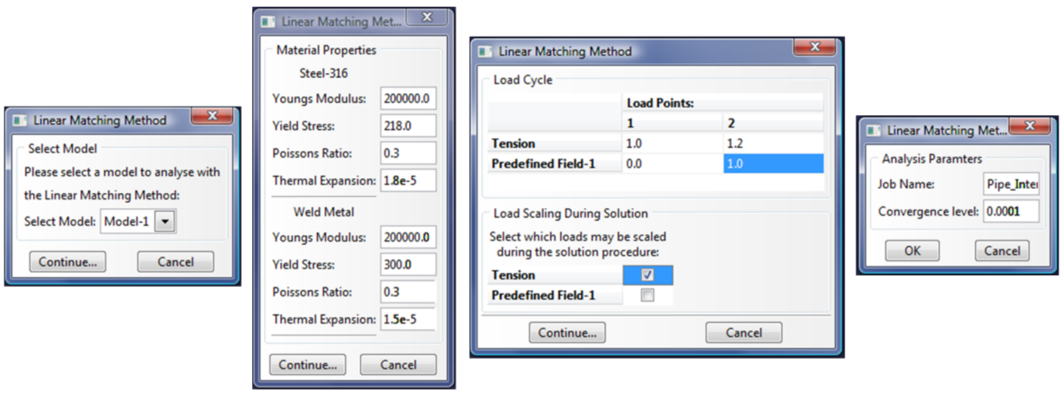

Figure 4. LMM Dialog Boxes

At each stage of the plug-in the data and options selected are checked to ensure no errors or invalid choices are being made by the user and to guide them on the correct use of the process. For example, one of the basic checks performed on the values of the material properties ensures non-negative values, which could result in the user seeing the warning shown in Figure 5.

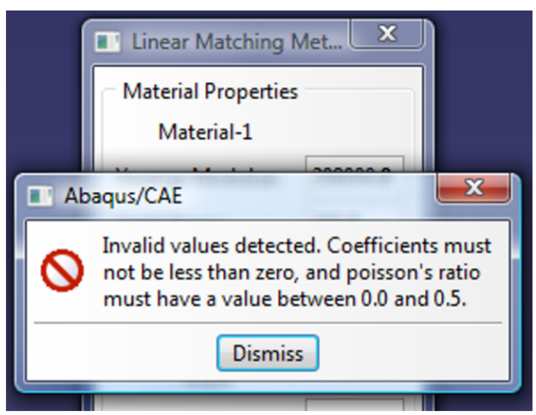

Figure 5. Example Warning of Incorrectly Entered Values

When the user has filled in all dialog boxes and clicks 'OK' in the final box, only then is the model modified for the LMM analysis. These modifications include:

- Creation of LMM analysis steps in line with the structure of the subroutines - One elastic analysis step per applied load, one null step and one LMM analysis step
- Creation of user materials with the correct number of Depvar
- Output requests
- Keyword block editing to request that the output is written to the Energy file (i.e. the .fil file)
- Creation of an Analysis Job including the LMM subroutines

All of these processes are performed in such a way so that the original model is entirely recoverable. For example when creating the user materials, the original materials are retained and the appropriate section definition is updated to the new LMM material. The only other input required from the user is to define the desired number of CPUs, allocated memory and any of the other options available and then submit the job. The analysis information not contained within the model itself (the convergence tolerance for example) is written to a formatted text file which is then read by the UEXTERNALDB subroutine when the analysis commences.

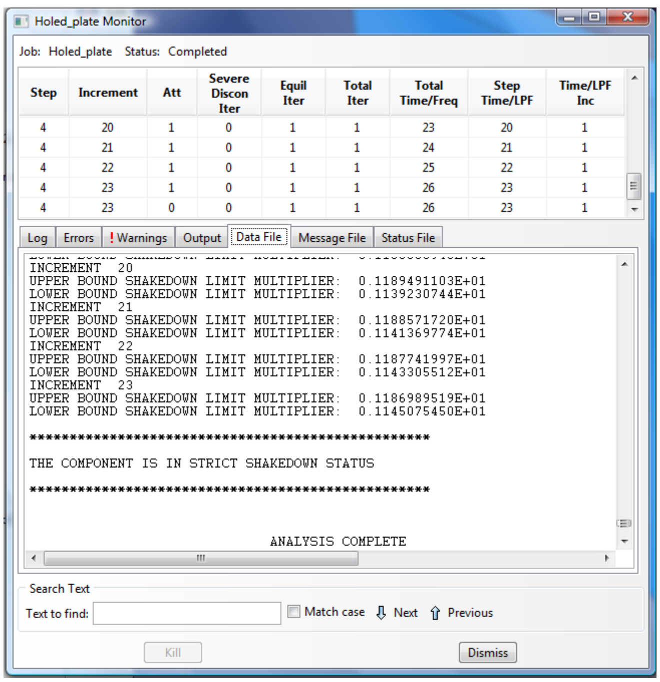

Figure 6. Monitor Dialog Box with LMM Printout

Whilst solving, the user can monitor the progress in the same way as any other Abaqus job. The plug-in prints the load multipliers in the Data file and this also appears in the 'Data file' tab of the 'Monitor' dialog box (see figure 6). The shakedown status of the component is printed along with the load multipliers (which give the user an indication of how well the solution has converged).

The development of the plug-in is still in the early stages, but is an example of how a user interface and process automation can be introduced with no previous knowledge of python code or the Abaqus GUI framework. Progress has been helped enormously by the presence of extensive help files and the process automation portal (http://www.abaqus.com/paportal), an online community where scripts and plug-ins are shared.

4.1 Plug-in Future Work

At present the functionality is limited to strict shakedown analysis so that a robust plug-in structure and analysis procedure can be created. When comprehensively tested, this structure will be extended to include more features (such as temperature dependent material properties and options for creep analysis) and the analysis of global shakedown.

5. Conclusion

The Linear Matching Method is a direct method capable of providing accurate shakedown calculations with options to aid structural integrity calculations. Where the pessimism of the elastic route in R5 is not acceptable the LMM, when implemented through a plug-in in Abaqus CAE, provides a superior alternative to cyclic inelastic FEA. This paper has presented a brief introduction to the method, the structure of the subroutines and outlined the current work of the plug-in which will provide the user interface to the method. Completion of this project will make the LMM process more accessible, thus allowing a more widespread use for structural integrity calculations in industry.

6. References

1. Ainsworth, R. A. (Ed.), "R5: Assessment Procedure for the High Temperature Response of Structures", Issue 3, British Energy Generation Ltd, 2003.
2. Bree, J., "Elastic-Plastic Behaviour of Thin Tubes Subjected to Internal Pressure and Intermittent High-Heat Fluxes with Application to Fast-Nuclear-Reactor Fuel Elements", Journal of Strain Analysis, Vol 2, no.3, pp 226-238, 1967.
3. Chen, H. F., Chen, W., Ure, J., "A Direct Method on the Evaluation of Cyclic Behaviour with Creep Effect", Proc. ASME PVP Conference, PVP2012-78065, 2012.
4. Chen, H. F., Ponter, A. R. S., "Shakedown and Limit Analyses for 3D Structures using the Linear Matching Method", International Journal of Pressure Vessels and Piping, 78, pp. 443-451, 2001.
5. Chen, H. F., Ponter, A. R. S., "A Direct Method on the Evaluation of Ratchet Limit", ASME Journal of Pressure Vessel Technology, 132, p.041202, 2010.
6. Chen, H. F., Ponter, A. R. S., Ainsworth, R. A., "The Linear Matching Method Applied to the High Temperature Life Integrity of Structures. Part 1. Assessments Involving Constant Residual Stress Fields", International Journal of Pressure Vessels and Piping, 83, pp. 123-135, 2006a.
7. Chen, H. F., Ponter, A. R. S., Ainsworth, R. A., "The Linear Matching Method Applied to the High Temperature Life Integrity of Structures. Part 2. Assessments Beyond Shakedown

Involving Changing Residual Stress Fields", International Journal of Pressure Vessels and Piping, 83, pp. 136-147, 2006b.

8. Koiter, W. T., "General Theorems for Elastic Plastic Solids" In: "Sneddon, J. N., Hill, R., (Eds.), "Progress in Solid Mechanics", North Holland, Amsterdam, pp.167-221, 1960.
9. Melan, E., "Theorie statisch unbestimmter systeme aus ideal-plastichem baustoff" Sitzungsber. d. Akad. d. Wiss. Wien 2A (145), 195-218, 1936.
10. Abaqus Process Automation Portal, http://www.abaqus.com/paportal
11. Tipping, D. J., "The Linear Matching Method: A Guide to the ABAQUS User Subroutines", Report E/REP/BBGB/0017/GEN/07, British Energy Generation Ltd, 2008.
12. Ure, J., Chen, H., Tipping, D. J., "Calculation of a Lower Bound Ratchet Limit Part 2 - Application to a Pipe Intersection with Dissimilar Material Join", European Journal of Mechanics A/Solids, In Press Corrected Proof, Available Online 18 April 2012.

7. Appendix - LMM Subroutine Flow Diagram

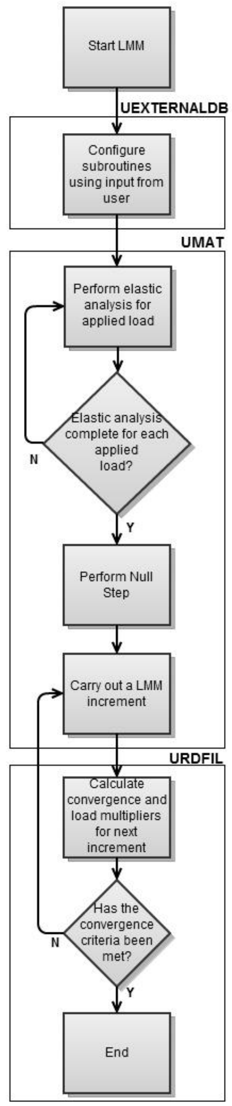

Figure A1. LMM Subroutine Flow Chart

8. Acknowledgements

The authors gratefully acknowledge the support of the Engineering and Physical Sciences Research Council and EDF Energy for their support during this project.