ISSN(Online): 2319-8753
ISSN (Print): 2347-6710

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 7, Issue 1, January 2018

Thermo-Mechanical Simulation Model Development on Residual Stresses in Rolled Aluminium Alloy

Purnendu Mondal[1*] and Christophe Pinna[2]

School of Architecture, Computing and Engineering, University of East London (Docklands Campus), London, UK [1]

Department of Mechanical Engineering, The University of Sheffield, UK [2]

ABSTRACT: The goal of this project is to investigate the Residual Stress in hot rolled aluminium alloys where both experiments and computer simulation will be implemented in order to get highest degree of investigation. This project is a miniature investigation process of real problem faced by AIRBUS, in the field of failure of wings against superfluous residual stresses. The residual stress level achieved from simulation matches to what received from the hole-drilling measurement after quenching process, hence, the necessity of the complex and expensive hole-drilling method can simply be omitted in the real industry resulting huge time saving in mass production units.

KEYWORDS: Thermo-mechanical, Simulation, Residual stress, Aluminium alloy, Non-Linear analysis, ABAQUS, CAE, Hot rolling, Quenching, Heat treatment.

I. INTRODUCTION

Most of the aviation industries utilize rolled aluminium alloy to a great extent for manufacturing the wings of aircraft. The wings made through the process of hot rolling usually suffer from having a tendency to warp and distort after they are cut from the large billet and assigned the desired shape. This is because of inherent residual stress generation during the prior manufacturing processes like rolling and quenching.

Residual stress measurement techniques sometimes help to estimate the possible stress level into which a component is subjected to service, though in many cases unexpected and prior failure occurs because of the presence of residual stresses combined with the service stresses which seriously shorten the life of components. Prediction of propagation and distribution of residual stresses in hot rolled aluminium alloy is of great importance in the industry, especially, in the manufacturing units [1-6]. Both the fields of hot rolling and machining can be investigated thoroughly to modify the aluminium rolling procedure to minimize stress development as low as possible.

Residual stress can be generated in a specimen due to several reasons like machining, assembly, welding, forging, heat treatment etc and can cause failure without prior notice. It can cause distortion, buckling in the material as well crack growth and fatigue life decrement. So, in the modern industry it is always desirable to avoid the generation of residual stress inside the component except some special cases.

For monitoring residual stresses inside a component, apart from hole-drilling with strain gages and ABAQUS simulation, there are several technologies, available in the industry like curvature method, compliance method, x-ray, neutron, electron methods, thermo-elastic, photo-elastic and piezoelectric methods etc.

Measurement of residual stresses is a globally accepted methodology to avoid the premature failure in components by determining stress propagation path in components. So, the technique arising from this project using CAE

DOI:10.15680/IJIRSET.2018.0701107

simulation and hole drilling method, can be implemented in various industries like automobile, heavy engineering, screening and crushing, aerospace, biomedical etc.

The technology used in this project will help a lot the manufacturers to get a clear idea how to minimize the residual stress level in alloys by improving manufacturing process and machining and hence, maximizing the life of their product without early failure. In many research laboratories and development centres, the procedures for measuring residual stress are being investigated using either experimental method or using only computer simulation [7-10]. But, hardly the combination of experimental measurement of residual stress and simulation through FE modelling in ABAQUS, been carried out together.

Novelty of the computer based simulation is that the residual stress level can be identified throughout inside the component, thus complexity and expenditure for the strain measurement techniques can simply be avoided. So, this unique analysis not only can bring a completely new technology that may drastically change the idea of residual stress measurement, but also can provide solution on how to constrain residual stress in the structure to its minimum level.

II. RESULT OBTAINED FROM LABORATORY EXPERIMENT OF HOT ROLLING AND HOLEDRILLING

The data are not processed by smoothing and these only reflect the original measured strains distributed along the depth [11-13].

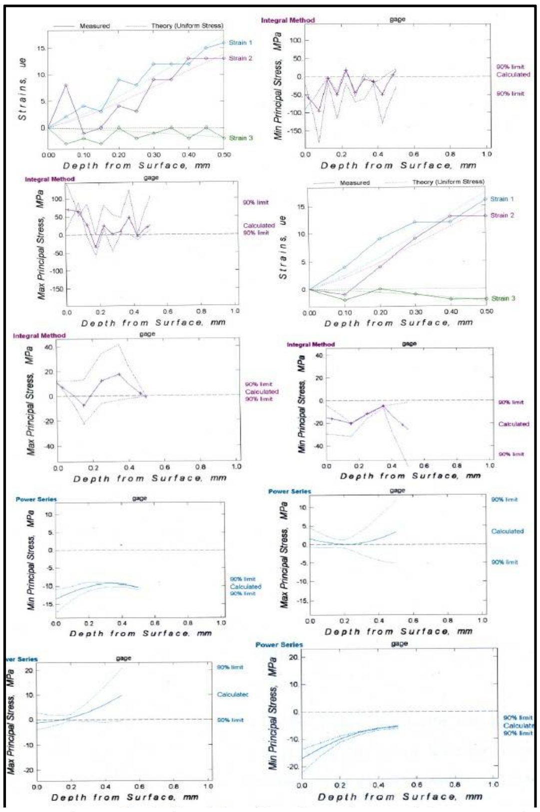


Fig. 1.(From left to right and top to bottom order) Strain chart (10 increments), Minimum principal stress profile with integral method (10 increments), Maximum principal stress profile with integral method (10 increments), Strain chart (5 increments), Maximum principal stress profile with integral method (5 increments), Minimum principal stress profile with integral method (5 increments), Minimum principal stress profile with power series method (10 increments), Maximum principal stress profile with power series method (10 increments), Maximum principal stress profile with power series method (5 increments), Minimum principal stress profile with power series method (5 increments).

From the results (Fig. 1), it is observed that in case of power series method, 10 increments and 5 increments yield almost the same results. It is because of the reason that power series method is less sensitive than the integral method to the effect of experimental errors. So, to be precise, results of 5 increment integral method are selected for comparing with finite element simulations.

III. SIMULATION OF HOT ROLLING

The numerical simulation of the hot rolling of aluminium alloy is developed using an implicit non-linear approach in ABAQUS/Standard version 6.5.1. In this analysis a coupled thermal stress condition along with a coulomb friction model is represented to achieve the contact situation.

The code used for this simulation is ABAQUS/ Standard implicit time integrator, which means that the time step is not limited by stability requirements [14,15]. Major disadvantage is that a nonlinear system equation must be solved at each time increment. At the same time advantage is that it is straightforward to generate a robust algorithm that varies the time increment at each time step to control local accuracy. As a result, numbers of steps are automatically minimized within the prescribed accuracy.

ABAQUS uses the Newton-Raphson method to solve the non-linear problems as a non-linear problem cannot be solved by a single system of equations. In the case of rolling simulation, load is applied by means of displacement of the roll which simply drags the stock inside the rolling pass because of the friction provided. So, the code breaks the whole procedure into a number of load increments and finds the approximate equilibrium configuration at the end of each load increment. The sum of all the incremental responses is the approximate solution of a non-linear system.

Initial Hand Calculations

Angular velocity of the roll = 10 RPM So, Angular Velocity = 1.0471 Rad/s And, the Radius of roll = 70 mm
So, Linear velocity= 73.304 mm/s

Due to the symmetry, only one half of the physical system is modelled in FEA, i.e. single roller with half of the stock thickness (Fig. 2).

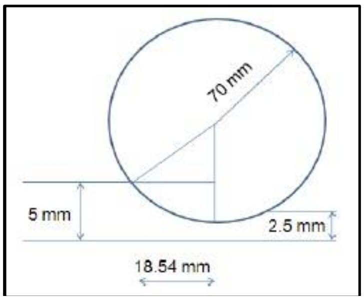
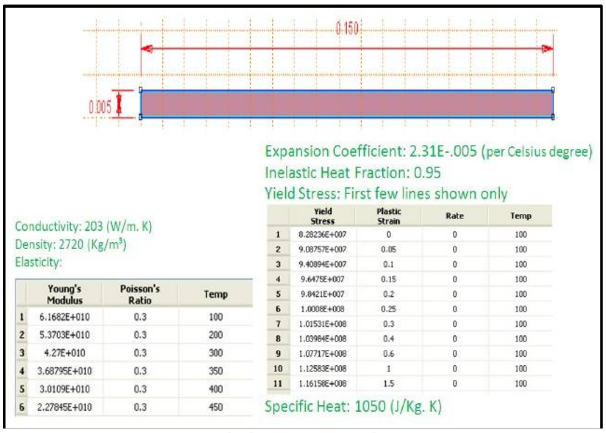


Fig. 2. Hand calculations for initial position of stock.

So, Initial Thickness= 5 mm
Reduction in thickness= 2.5 mm
So, from the calculation l= 18.54 mm
Length of the stock= 150 mm
Total Passing Length= 168.54 mm
Running time= 168.54/ 73.304 sec = 2.2992 mm

Parts, Properties and Assembly

Properties of material which undergoes the rolling process have been defined with respect to different temperatures and strain rates (Fig. 3).



Expansion Coefficient: 2.31E-.005 (per Celsius degree)
Inelastic Heat Fraction: 0.95
Yield Stress: First few lines shown only

	Yield Stress	Plastic Strain	Rate	Temp
1	8.28236E+007	0	0	100
2	9.08757E+007	0.05	0	100
3	9.40894E+007	0.1	0	100
4	9.6475E+007	0.15	0	100
5	9.8421E+007	0.2	0	100
6	1.0008E+008	0.25	0	100
7	1.01531E+008	0.3	0	100
8	1.03984E+008	0.4	0	100
9	1.07717E+008	0.6	0	100
10	1.12583E+008	1	0	100
11	1.16158E+008	1.5	0	100

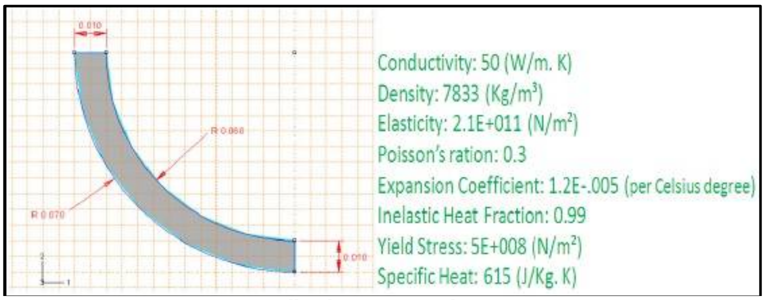
Conductivity: 203 (W/m. K)
Density: 2720 (Kg/m^3)
Elasticity:

	Young's Modulus	Poisson's Ratio	Temp
1	6.1682E+010	0.3	100
2	5.3703E+010	0.3	200
3	4.27E+010	0.3	300
4	3.68795E+010	0.3	350
5	3.0109E+010	0.3	400
6	2.27845E+010	0.3	450

Specific Heat: 1050 (J/Kg. K)

Fig. 3. Stock dimensions and properties.

Both the roller and stock have been defined as deformable bodies i.e. the real scenario. But, to put the proper constraining and boundary condition it is essential to put a rigid surface attached to the inner roller surface. This is one sort of special technique to run the simulation effectively. So, in the assembly ideally three parts are present, deformable roller, deformable stock and rigid surface (Figs. 4 and 5). The roller is considered as hollow one with wall thickness of 10 mm.



Conductivity: 50 (W/m. K)
Density: 7833 (Kg/m^3)
Elasticity: 2.1E+011 (N/m^2)
Poisson's ration: 0.3
Expansion Coefficient: 1.2E-.005 (per Celsius degree)
Inelastic Heat Fraction: 0.99
Yield Stress: 5E+008 (N/m^2)
Specific Heat: 615 (J/Kg. K)

Fig. 4. Roller dimensions and properties.

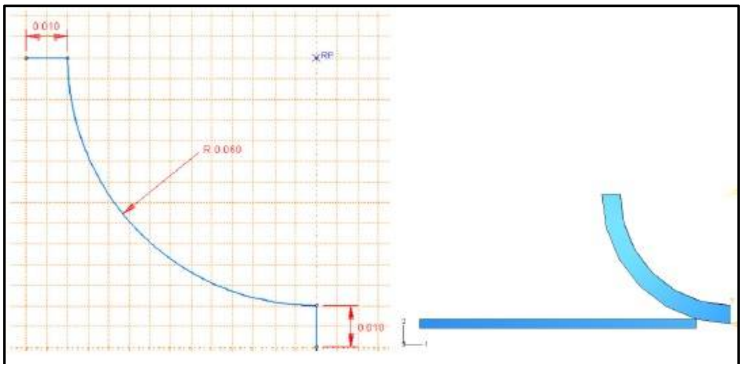


Fig. 5. Rigid surface attached and assembly.

The initial position of the stock with respect to the roller is obtained from the hand calculations. The material behaviour of the stock is modelled using inverse hyperbolic sine functions which yield a visco-plastic behaviour linking the flow stress with temperature, strain and strain rate (Fig. 6).

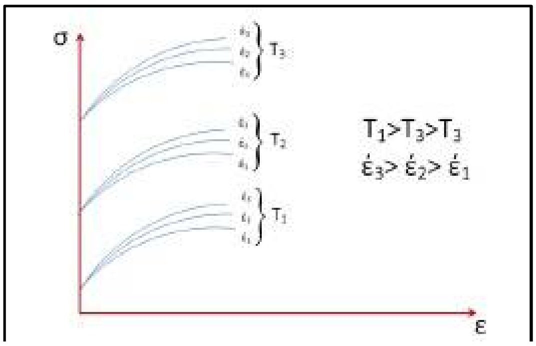


Fig. 6. Type of relationship used between stress, strain and strain rate.

Steps, Sets and Surfaces

There are two steps defined in this simulation. One is to establish the contact between the stock and the roller and the other is rolling pass. In case of rolling pass, simulation is carried out up to the point where the stock is reaching to its steady state condition in order to reduce the simulation time and hence, the computational time and memory usage [16-19].

Sets are defined in order to define the boundary conditions. For angular velocity provided to roll, initial feed to the stock and constraining the plate and roll as per real conditions (Figs. 7 and 8).

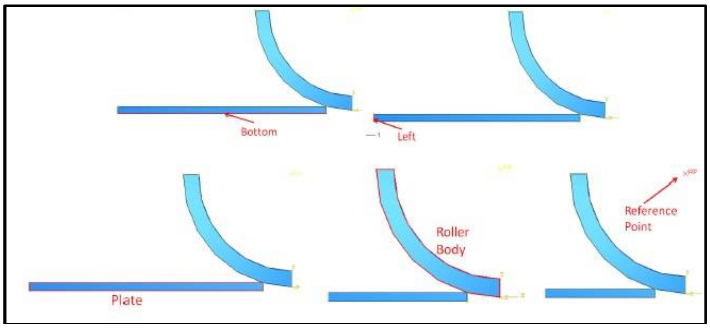


Fig. 7. Sets defined in simulation.

Surfaces are defined to put the interaction in the computer simulation.

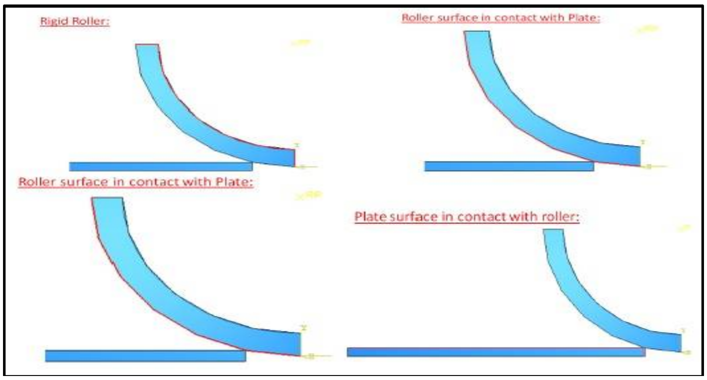


Fig. 8. Surfaces defined in red colour.

As this analysis is non-linear one, a single step is broken down into smaller increments so that another solution path can be followed. At the beginning of the each step, it chooses the amount of load in terms of displacement to apply within first increment as a percentage of the total load applied in the whole step. After the first step ABAQUSautomaticallyselectst size of the subsequent increments according to the us indication, in which simulation is the temperature change p increment per node allowed up to 300°C. At the end of ea increment the structure is in equilibrium and ready for the ne increment.

A non-linear structure responses to a small load increment ΔP and ABAQUS calculates the initial stiffness matrix K_0 based on configuration u_0. ΔP also calculates a displacement correction C_a which updates the system configurations (Fig. 9).

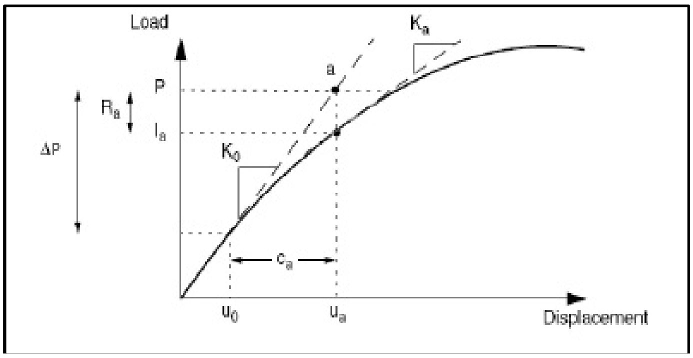


Fig. 9. First iteration in an increment in ABAQUS.

ABAQUS then creates a new stiffness matrix K_a and the whole procedure repeated until the system converges. K_a calculates the structures internal force I_a and if P is the total applied load, then R_a (force residual) is calculated as difference between P and I_a.

So, $R_a = P - I_a$.. (1)

If the solution does not converge, ABAQUS performs another iteration to balance the internal and external force. Second iteration uses the stiffness matrix K_a to determine another displacement correction c_b which tries to bring the system in equilibrium (Fig.10).

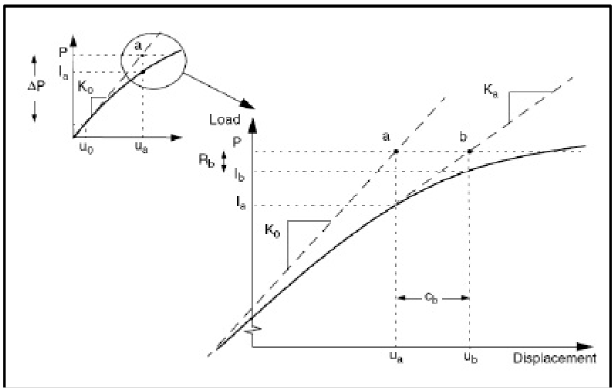


Fig. 10. Second iteration in an increment in ABAQUS.

A new force residual R_b is calculated once again using the internal force and configuration of system again updated to

u_b. If necessary, further iterations are performed.
The convergence of the solution is affected by:

- Relative position of stock androll.
- Load applied in the first increment of eachstep.
- Type of element used or in other words aspect ratio due to deformation.

In the rolling pass (2nd step), increment is mentioned up to 1000 while it is taking 463 to get solved.

Interaction, Boundary Conditions and Loads

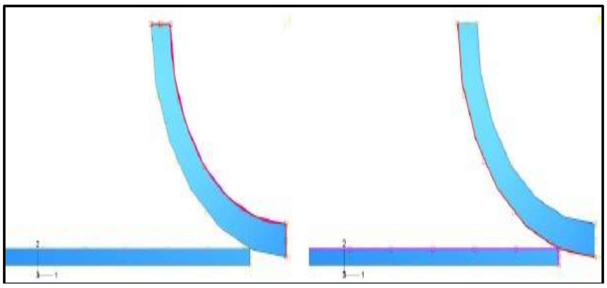


Fig. 11. Contact interactions.

Stock and roll are in contact with each other and the contact pressure could be very high as the reduction ratio is 50%. There are two interactions defined to establish the contact. Fist contact is between the rigid surface and the deformable roller. This is a type of rough contact without any sort of penalty. The second one is between the roller andstock. In this contact phenomenon, some key parameters to be definedaccuratelytodescribetheconditionswithinthecontact area regarding heat transfer, friction and shear stress. So, friction coefficient is defined as 0.3 and at zero clearance conductivity is put as 48000 while zero conductance is put for 1E-12 clearance value (Fig. 11).

The standard Coulomb friction model assumes that no relative motion occurs if the equivalent frictional stress τeq is less than the critical stress defined as:

$$\tau cr= \mu P_C \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \text{(2)}$$

Here, P_C is the contact pressure and μ is the friction coefficient.

ABAQUS/ Standard employs an extension of this classical isotropic Coulomb friction model that includes an additional limit on the allowable shear stress τmax so that

$$\tau cr = \min(\mu P_C , \tau max) \ldots\ldots\ldots\ldots\ldots\ldots \text{(3)}$$

Whenever the equivalent stress is at the critical stress or over, ABAQUS allows slip to occur between the surfaces. Frictional coefficient between roller and the stock is chosen as 0.3 which is quite appropriate from the theoretical point of view. Frictional coefficient also varies with a lot of other criteria like reduction ratio, temperature and rolling speed. Though it varies with these factors, still, it is convenient to consider a constant value for friction coefficient. It is the reason, why 0.3 is selected as this coefficient.

Middle plane of the stock is kept as fixed in vertical direction while center of the roll to be fixed both in horizontal and vertical directions. Left side of the plate is given the feed as per hand calculations and roller is given calculated angular velocity (Fig. 12).

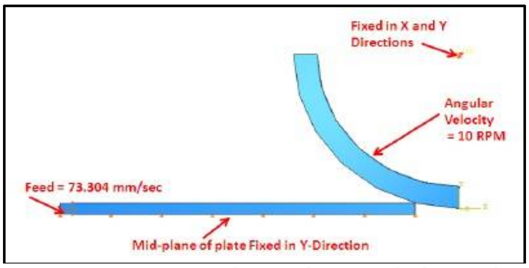


Fig. 12. Boundary conditions applied on stock.

As this analysis is of coupled thermal displacement type, initial temperatures of stock and the roller are put as 400°C and 20°C respectively. In order to have accurate heat flow from the specimen, material behaviours have been put with highestaccuracy.

Elements

Element type chosen for both the stock and roller is CPE4RT, which is compatible with 4-node plane strain thermally coupled quadrilateral, bilinear displacement, temperature and reduced integration. 150 mm long x 5 mm height stock is consisting of 300 numbers of elements in horizontal direction and 10 numbers of elements in vertical direction. As application of adaptive mesh control yields problems in solving this simulation, choice of initial level of meshing is vital one for this simulation. Finer meshing density is used for stock than that of roller (Fig.13).

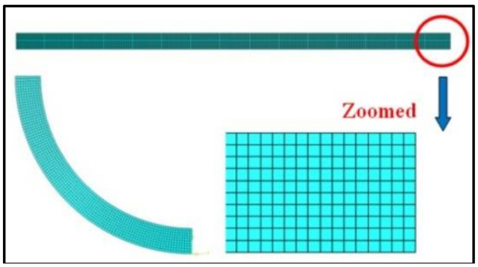


Fig. 13. Meshing of roller and stock

Results

Main focus of obtaining the resultsis to get the output from steadystateconditions.VonMisesstress,strain,strainrateand temperature have been taken under considerations only after judging that the steady state condition achieved well. Though the running time in the real stock length is 2.2992 seconds, only after 0.8 seconds, this simulation is reaching to a stage which can be considered as steady one. This much of simulation takes around four hours to getsolved (Fig.14).

Stresses

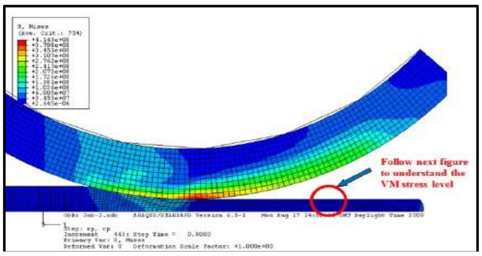


Fig. 14. Von-misesstress contour.

From the stress profile ofthe stock, it is understood that the rolled stock has been achieved the steady state as the value of the stresses are unique throughout the surrounding length of thestock (Fig. 15).

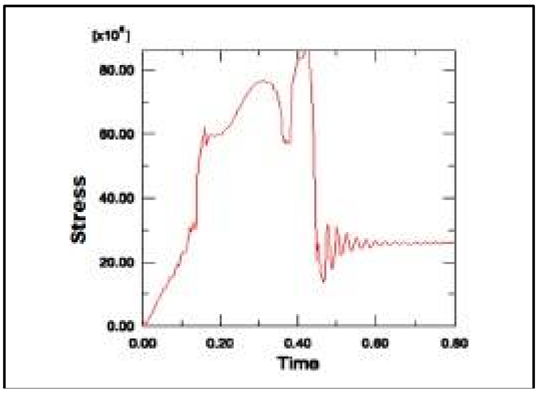


Fig. 15. Von-misesstress level with respect to time for selected elements (encircled).

Logarithmic Strains

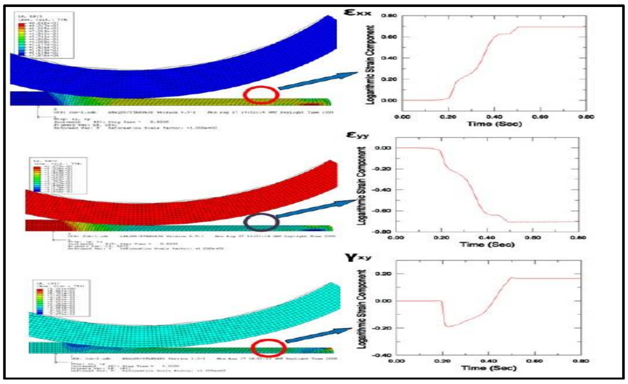


Fig. 16. Three logarithmic strain components plotted in contour and graph.

From both the contours and graphs, it is clear that the portion of stock of interest has been reached to steady state condition (Fig.16). Also, incompressibility of the material is being proved by:

$$\varepsilon_{xx} + \varepsilon_{yy} = 0 \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (4)$$

Strain Rate

Equivalent strain rates are available in ABAQUS/ implicit and defined as:

$$\acute{\varepsilon}_{eq} = \sqrt{}(\acute{\varepsilon}_x + \acute{\varepsilon}_z) \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (5)$$

$\acute{\varepsilon}_x$ and $\acute{\varepsilon}_z$ are maximum and minimum principal strain rates (Figs.17 and 18).

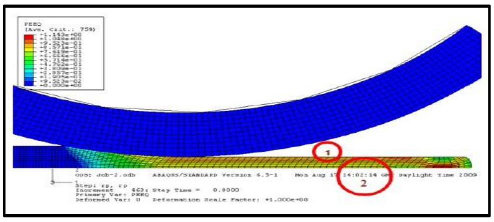


Fig. 17. Equivalent strain rate in top and middle plane.

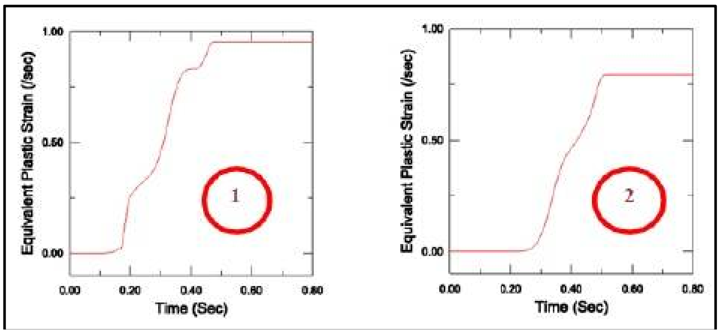


Fig. 18. Equivalent strain rate in top and middle plane.

Temperature

It is found that the steady state temperature is 233°C (Fig.19).

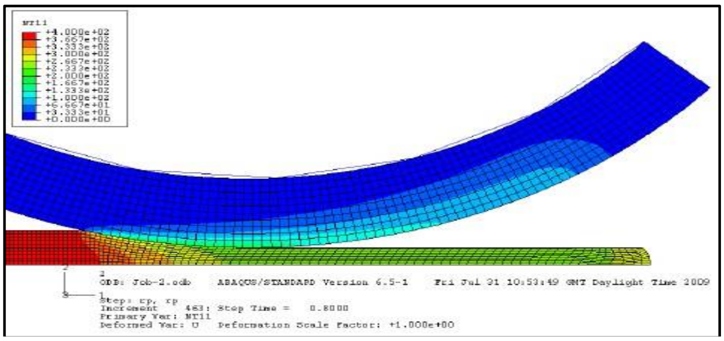


Fig. 19. Temperature distribution after the single pass rolling.

Discussion

All the stress, strain, strain rate and thermal components are successfully obtained from the hot rolling simulation. So, coupled thermal displacement analysis is perfectly carried out without any drawback. Stress, strains and temperaturesare coming as per the expectation from the theoretical point of view. So, this simulation can be said as successful in as aspects and the steady state data can be transferred to the initial level of quenching simulation.

IV. SIMULATION OF WATER QUENCHING

Water is most popular and effective quenching medium used for hot aluminium alloys. Main advantages are that water is cheap, readily available and reasonably flexible with the temperature change. Major technical advantage is that it can provide rapid quenching for achieving high properties in many different alloys. It is always convenient to use the water at room temperature but the main drawback of this method is that many portion of the specimen can wrap or distort during rapid quenching which need further straightening procedure. Also, the thicker plate may achieve higher level of residual stresses.

In the water quenching process, total heat transfer between water and specimen surface can be described by the governing equation composed of film coefficient and temperature difference.

$$q = -h\ (\theta_{wall} - \theta_{fluid}) \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \ (6)$$

How the final temperature is obtained, can be calculated through the following numerical equation:

$$dT/dr = h.\ A.\ (\theta_{wall} - \theta_{fluid})/\ V.\ \rho(T).\ c_p(T) \ldots\ldots\ldots\ldots\ldots\ldots\ldots \ (7)$$

Both the film coefficient and the ambient temperature are taken into consideration by ABAQUS standard and are considered as constant over the steps used in coupled thermal displacement analysis.

ABAQUS Modelling

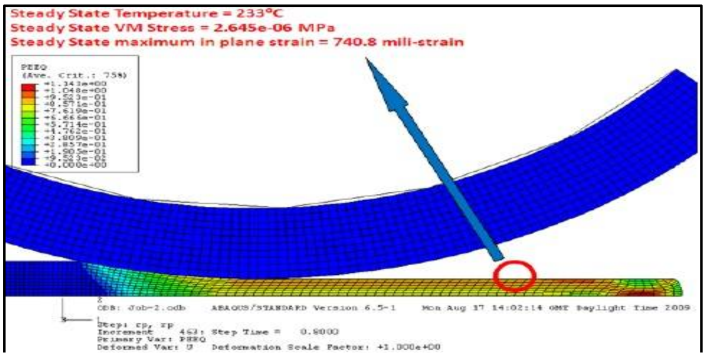


Fig. 20. Steady state conditions taken from rolling simulation.

Main aim of the finite element modelling is to put the stress, strain and temperature data of the steady state condition of hot rolling simulation, as the input data for quenching simulation for all the elements (Fig. 20).

During this simulation some problems were faced while trying to model the problem using database application menus. So, this simulation is finally generated by writing scripts in ABAQUS. The stock is the main part needed to take under consideration for further analysis and it is modelled by using the same structure and dimensions without considering symmetry (150 long × 5 mm thick). Properties are kept the same used in the rolling simulation and only a single step has been used. In the interaction module, film coefficient valuehas been set to 50 $W/m^2.°C$ for all the surfaces. Element type has been selected the same as in rolling one (CPE4RT) to achieve the coupled thermal displacement analysis. Another special consideration to carry out the coupled analysis is that lower end points of the stock have been fixed in the 2-D spaceframe,i.e. linear displacements in X and Y direction and rotational displacement about Z axis are zero.

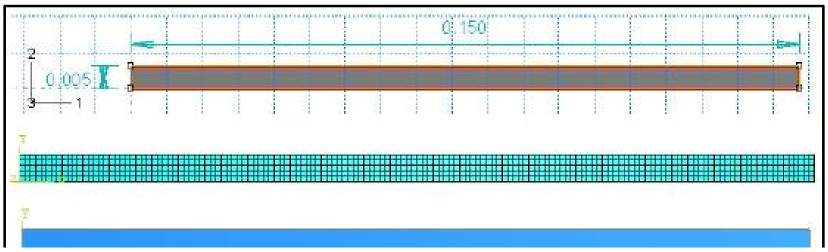


Fig. 21. Sketch, meshing and boundary conditions put in the stock (U1, U2, UR3 = 0).

As per the real scenario this simulation should take more than hundred hours to get solved (Fig. 21). So, in order to reduce the simulation time without reducing the accuracy of the result film coefficient has been selected as 5000 W/m^2.°C instead of 50 W/m^2.°C.

Results

As the film coefficient is put 10 times more than the real one in order to obtain the result in faster manner, only 65.5 seconds is good enough to get the perfect results. Still, with this changed value, simulation took around nine hours to visualize the result.

Temperature

Temperatures profiles mentioned in Figs. 22 and 23.

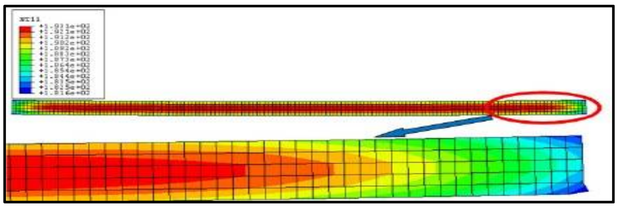


Fig. 22. Intermediate temperature profile.

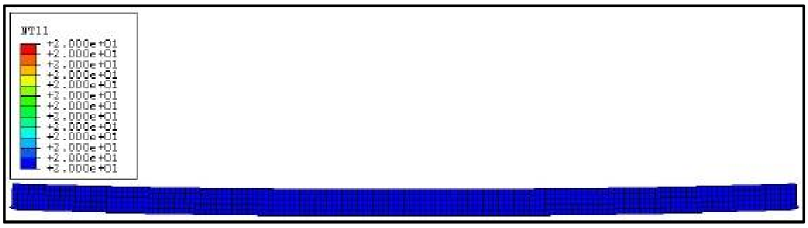


Fig. 23. Final temperature profile.

Stress

Stresses generation throughout the specimen (Fig. 24).

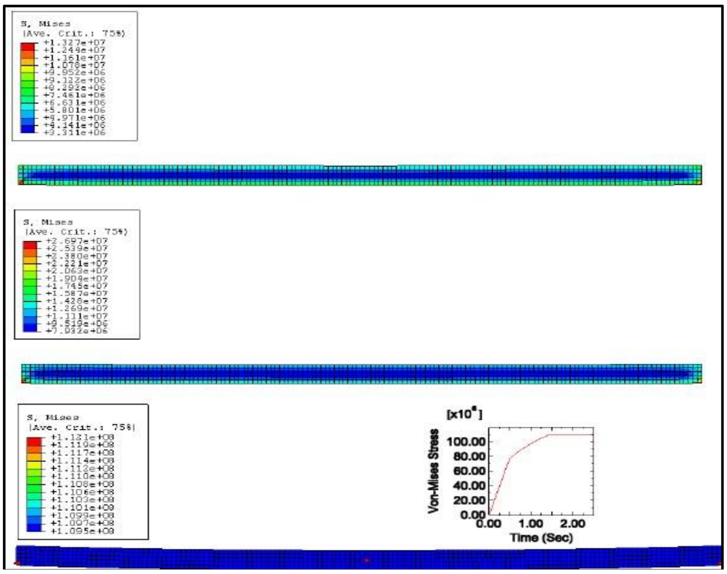


Fig. 24. Von Misesstresses generation throughout the specimen (all three in chronological order from top to bottom) and its changes with respect to time at the absolute center of specimen (marked red).

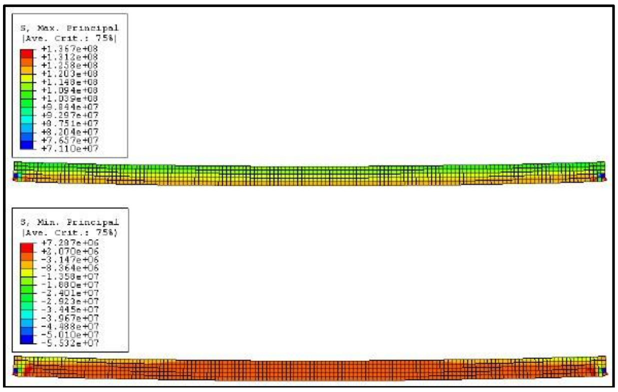


Fig. 25. Maximum and minimum principal stress finally obtained.

To obtain the coupled outputs, mechanical constrains are put at the bottom two corners (Fig. 25). In real it is not feasible, but, this is the best consideration to run this simulation. This is the reason why stress generated at the bottom two corners and the stock is bent. These two factor can simple be ignored as main area of focus is stress, strain, strain rate and temperature of the overall specimen.

Strain

Maximum and minimum in plane logarithmic strain (Fig. 26).

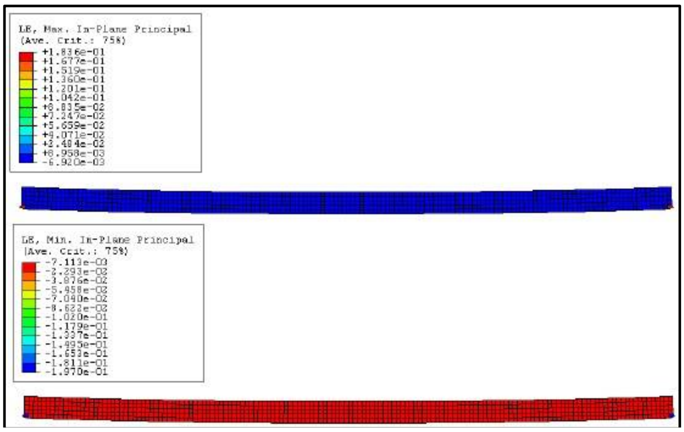


Fig. 26. Maximum and minimum in plane logarithmic strain.

Discussion

Stress, component of strains and temperature profiles are coming as expected from the theoretical point of view. It is understood that the final temperature after the quenching will be 20°C and the simulations are yielding this value, too. More over intermediate profiles for all the criteria are absolutely perfect with respect to time.

V. COMPARISON BETWEEN EXPERIMENTAL RESULTS AND SIMULATION OUTPUT

The experimental data for comparison taken from integral method with 5 increments and the parameters to be compared are maximum and minimum principal stresses, strains and temperature. But in the experiment, data received for the strain is the direct readings from the instrument. So, in order to compare the results, these strain values need to be transferred to the maximum and minimum principal strains. The strain values received:

$$\varepsilon_a = 16.0\ \mu s,\ \varepsilon_b = 12.5\ \mu s,\ \varepsilon_c = -3.0\ \mu s$$

So, by the formula, maximum and minimum principal stresses are given by:

$$\varepsilon_{1,2} = 0.5\ [(\varepsilon_a + \varepsilon_c) \pm \{(\varepsilon_a - \varepsilon_c)^2 + (2\varepsilon_b - \varepsilon_a - \varepsilon_c)^2\}]^{0.5} \ ....\ (8)$$

Putting the above values of strains in the above equation,

$$\varepsilon_{1,2} = 17.74,\ -4.74\ \mu s$$

So, based on experimental data obtained and simulation results from both the ABAQUS modellings, a chart of comparison is prepared (Table 1):

Table 1. Comparison of results obtained.

	Experiment	Theory	ABAQUS simulation
Maximum principal stress (MPa)	18	44.0	98.4
Minimum principal stress (MPa)	-28.8	-60.0	-3.14
Maximum plane stress (μS)	17.74	17.0	8.96
Minimum plane stress (μS)	-4.74	-3.0	-7.11
Temperature (°C)	20.0	N/A	20.0

VI. CONCLUSION

Hot rolling simulation in this project is the most successful one. All the outputs are greatly matched to the simulation of “Hot rolling of Aluminium Alloy” carried out by a group of researchers in “Institute of Microstructural and Mechanical Process Engineering: The University of Sheffield”. At the same time all the outputs are as per the hypothetical prediction. As long as temperature is main focus area for quenching analysis, the finite element modelling in ABAQUS/Standard yields good prediction. Moreover, components of final stresses and strains are also as per expectation of theoretical views.

Some variations have come at the end of seconds imulation, mainly because of there dundantmechanicalconstrainsputinthequenchingmodellinginordertorunaperfect coupled thermal displacement analysis.Fromthepracticalpointofviewitisunderstoodthatthestockusedinquenching is acting like a thin plate fixed at both theends.So,naturally,thestresslevelswillbeabithigherthanthetheoretical and experimental results. It is same reasonwhythesimulated strain levels differ from the otherresults.Anotherpossible reason may be the scaling up of filmcoefficientusedin quenching simulation to get the resultswithinpermissiblelimit of time. In the hot rolling simulation,accuratetimevaryingfrictioncoefficientscanalsohigherthelevelofprecisioninthefinalresults.Also,inthequenchingsimulation,elementsizehasbeenchosenas1mm×1mm.So, as per the theory, Gaussian point is coming at 0.5mmfromthe top surface for the first row of elements. But therealvalueof stress in that particular element may be atsomeotherheight. So, refining the mesh size can result a betterresultwithmore accuracy. Some other reasons forthedifferencesbetween results may be our considerationofperfectrectangularblockduringquenchingsimulation,whileinrealthestockgotcurvatureattheends.Mostimportantly,steadystateconditionsaretakenfromthehotrollingsimulationforquenching simulation and no heat lossisconsideredmeanwhile. But in actual case, all the elements of stockarenotreached in steady condition and there must be afinitetimeintervalbetweenhotrollingandquenching,resultingheatloss.So,furtherscopeofimprovingtheresultsofthisproject can be listed from the above conclusions.Morerealisticmechanicalconstraininginthequenchingsimulation,puttingmore accurate time varying coefficient of friction in therolling simulation, refining the mesh density and finally, enabling the finite element model to run as long as it requires by using non- magnified accurate value of film coefficient, can hugely enhance the accuracy of this results to match the experimental and theoretical ones. Considerations of real shape and actual states of all the elements after the hot rolling simulation and accounting the heat loss before the quenching simulation can also predict the results in a much better way.

Enormous scope to improve this project is ahead and well defined. Due to the time constrain, the research in finite element modelling is paused in this stage. But, it can be predicted that using and modifying the coding file generated in this project, finite element modelling with this technique will be known as the best method for measuring the level of residual stresses inside any structure in the nearest future.

REFERENCES

[1] C. Boldetti, C. Pinna, IC.Howard, G. Gutierre, “Experimental Measurement of the Deformation in Hot Rolling of Aluminium 1% Mn: The Grid Technique”, Applied Mechanics and Materials, vol.3 no. 4, pp. 111-116, 2006.

[2] X. Duan, T. Sheppard, “Three dimensional thermal mechanical coupled simulation during hot rolling of aluminium alloy3003”, International Journal of Mechanical Sciences, vol. 44, pp. 2155– 2172, 2002.

[3] F. Boitout, D. Dry, Y.Gooroochurn, P.Mourgue, H. Porzner,“Heat Treatment Simulation”, Paris: ESI Group, pp. 39-44, 2008.

[4] HT. Lee, WP. Rehbach, FC. Hsu, TY. Tai, E. Hsu, “The study of EDM hole-drilling method for measuring residual stress in SKD11 tool steel”, Journal of Materials Processing Technology, vol. 149, pp. 88–93, 2004.

[5] H.Pron, JF. Henry, B. Flan, J. Lu, S.Offermann, JL.Beaudoin, “Estimation of residual stresses induced by shot-peening”, Int JThermSci, vol. 41, pp. 369–375, 2002.

[6] O. Sicot, XL. Gong, A.Cherouat, J. Lu, “Influence of experimental parameters on determination of residual stress using the incremental hole-drilling method”, Composites Science and Technology, vol. 64, pp. 171–180, 2004.

[7] M. Honner, P.Lito, M.Svantner, “Thermography analyses of the hole-drilling residual stress measuring technique”, Infrared Physics & Technology, vol. 45, pp. 131–142, 2004.
[8] QX. Yang, M. Yao, J. Park, “Numerical simulation on residual stress distribution of hard-face-welded steel specimens with martensite transformation”, Materials Science and Engineering, A364, pp. 244–248,2004.
[9] W. Lehnert, ND.Cuong, “Experimental and mathematical simulation of microstructural evolution during hot rolling of A1 and Cu material”, Journal of Materials Processing Technology, vol. 60, pp. 567-574, 1996.
[10] AJ. Brand, S.Kalz, R. Kopp,“Microstructural simulation in hot rolling of aluminium alloys”, Computational Materials Science, vol. 7, pp. 242-246, 1996.
[11] PJ. Withers, HKDHBhadeshia, “Residual stress Part 1 – Measurement techniques”, Materials Science and Technology, 17, pp. 355-365, 2001.
[12] Vishay,“Measurement of Residual Stresses by the Hole-Drilling Strain Gage Method”, Micro-Measurements, 11053,pp. 19-33, 2010.
[13] P. Mondal, “Residual Stress in Rolled Aluminium Alloys”,The University of Sheffield, pp.1- 46, 2009.
[14] C.Boldetti, C.Pinna, Howard,G. Gutierrez, “Measurement of deformation gradients in hot rolling of AA3004”, Experimental Mechanics, vol. 45 no.6, pp. 517-525,2005.
[15] SY. Hilali, BJ. Wang, “ABAQUS Thermal Modeling for Electrical Assemblies”, ABAQUS User’s Conference, pp. 441-457, 1995.
[16] JC. Middleton, “Residual Stress Measurement at Corus”, Rotherham: Corus, Research, Development & Technology, pp. 5-8, 2001.
[17] GS. Schajer, “Hole-Drilling Residual Stress Measurements”, Proceedings of the SEM Annual Conference, 2009.
[18] X. Duan, “Some Problems In Hot Rolling Of Al-Alloys Solved By The Finite Element Method”, Bournemouth University, 2001.
[19] X. Che, “Computer Simulation of The Thermal- Mechanics of Steel Quenching”, MaterialsScience and Engineering, 1998.