Mechanical Stress Relief Simulation
David H. Johnson and Richard B. Englund
Penn State – Erie, Erie, PA, USA

Abstract
A method of producing a known residual stress is sought so that methods for measurement of residual stress may be verified and calibrated. Equally, a clear classroom demonstration of a situation which causes residual stress and then relief of that residual stress is desired. It is shown in this paper that a flat aluminum specimen may be prepared with stress concentration discontinuities, and the part loaded so that the average stress and strain is below the yield point and the maximum stress in the discontinuity is beyond the yield point. Upon unloading from this condition, residual stresses will be present in the part, which may then be released by sawing through the part. Classical predictions of residual stress are correlated to a finite element model which simulates the loading, unloading, and cutting steps of the process. Observations on the difficulties inherent in experimentally accomplishing this process are also presented.

Introduction
Residual stresses are often present in physical parts. Mechanical action on finished parts and manufacturing operations of many types leave residual stresses. Two common examples are cold rolling of steel sheet, and cooling welds. Cold rolling significantly yields part of the material, but not uniformly through the thickness. Hence in a machining operation where one surface is removed, the part will warp to some degree, often quite visibly. Welds cool non-uniformly, leaving the portion which cools last in tension and the part which cools more quickly in compression. Weld distortion is a common result. For any case, residual stresses are present in the part and all operating stresses are added to the residuals present in the part. Since residual stress can affect the performance or life of the part, determination of residual stress is sometimes necessary, and certainly an awareness of residual stress is valuable to the designer of mechanical systems.

Stress relief techniques are one method used for measurement of residual stress in mechanical parts. Generally the method requires placement of a strain gage on the part and then removing material so that some of the stress is unlocked, permitting the stress at the location of the gage to be somewhat relieved. This may be achieved by the hole drilling method1 where three gages in a single backing are affixed to the surface, a bridge balanced for each gage, and then the material removed in the center of the three gages. Residual compression will elongate the gages as the material is allowed to expand into the hole, and conversely residual tension will shorten a gage when the material is released by the hole. From the change in strain indicated by the three gages as drilling releases the stress, the stress state present at the location prior to drilling may be determined. One direction of residual stress may be determined by placement of a single gage and by sawing through the part to release any residual stresses perpendicular to the saw cut. Saw cutting destroys the part, but is readily accomplished with standard equipment and understood by most observers.

The intent of this present work is to design a specimen that may be reliably manufactured and a predictable residual stress induced into the part for release upon sawing. Such a specimen can be used for a classroom example or laboratory exercise, but may have application to calibration of residual stress measurement equipment if indeed the residual stress in the part is reliably produced. Additionally, the finite element model used to compliment the experiment allows students to visualize the stress-strain behavior in the specimen.
Analytical Development

Classical stress analysis techniques, combined with typical assumptions of elastic/perfectly plastic material behavior, were used to select a specimen size and configuration. It was desired that the specimen be small enough to be loaded essentially to the yield point by a moderate sized universal testing machine with 60 kip (267 kN) capacity. The material chosen for analysis is 6061-T6 aluminum so that the strains will be higher than steel, and so that the specimen size may be relatively large yet the required force still well below the assumed machine capacity. A specimen of 0.5 inch (12.7 mm) thickness and 2.4 inches (61 mm) width was chosen to stay within these limitations.

A semi-circular notch in each edge of a tension member was assumed, and a combination of notch radii and depth was selected resulting in a stress concentration of 2.05 approximately. The basic dimensions and geometrical stress concentration factor are shown in Figure 1. The tensile force required to bring the minimum section to the yield stress, and the strain at the yield stress are:

\[
\begin{align*}
P_{\text{yield}} &= \text{net area } \times S_y = .8 \text{in}^2 \times 40 \text{ksi} = 32 \text{kip} \quad (= 142 \text{kN}) \\
\epsilon_{\text{yield}} &\approx \frac{S_y}{E} = \frac{40 \text{ksi}}{10,000 \text{ksi}} = .004 \text{in/in}
\end{align*}
\]

![Figure 1. Basic specimen dimensions and stress concentration factor](image)

For this demonstration, it was decided to take the net section to 75% of the yield stress, or 30 ksi (207 MPa) average stress and .003 in/in (mm/mm) average strain, hence requiring a force of 24 kips (107 kN), equal to 75% of the 32 kips (142 kN) computed above. With the stress concentration factor of 2.05, the computed stress at the radius should be

\[
\sigma_{\text{max, theoretical}} = 2.05 \times 30 \text{ksi} = 61.5 \text{ksi} \quad (= 424 \text{MPa})
\]

\[
\epsilon_{\text{max}} = 2.05 \times .003 \text{in/in} = .00615 \text{in/in}
\]

The maximum stress is not attainable since the part will yield to keep the stress at no greater than the yield stress of 40ksi (276 MPa), but the strain at the radius will indeed be the approximately .0062in/in when the load is applied. The initial loaded conditions for both the radius and the center of the specimen are shown on Figure 2. Of the .0062 in/in of strain present at the radius, .0022in/in is plastic strain. When the load is
removed, the strain in the part will try to return to zero strain. Because the center of the part returns to zero strain along the original loading line and the radius returns to zero strain along a line with .0022 in/in of plastic strain, the radius will have a compressive residual stress and the center of the specimen will have a tensile residual stress. Based very loosely on the depth of the stress concentration computed theoretically for an infinite plate with a central hole, it was assumed by the second author that the stress concentration could be taken as linear and to reduce to zero by .050 inches (1.3 mm) of depth into the specimen. The residual stress and force in the specimen when the load is removed are estimated:

\[
\sigma_{\text{residual}} = \varepsilon_{\text{plastic}} \times E = .0022 \text{in/in} \times 10,000 \text{ksi} = 22 \text{ksi}
\]

\[
P_{\text{residual}} = 2 \text{ sides} \times .050 \text{in deep} \times .50 \text{ in wide} \times \frac{22 \text{ksi} + 0 \text{ksi}}{2} = .55 \text{kip}
\]

Figure 2. Stress-strain path for loading, unloading, and sawing
To offset this .55 kip (2.45 kN) compressive force in the radius, there must be a tensile stress in the central region of the specimen. The central region of the specimen is 1.60 in (2 × .05 in) = 1.50 inches (38 mm). The residual stress and strain in the central portion of the specimen is then

\[
\sigma_{\text{residual tensile}} = \frac{P_{\text{residual}}}{A_{\text{central}}} = \frac{.55 \text{kip}}{1.5 \text{in} \times .5 \text{in}} = .73 \text{ksi} \quad (= 5.03 \text{MPa})
\]

\[
\epsilon_{\text{residual tensile}} = \frac{\sigma_{\text{residual tensile}}}{E} = \frac{.73 \text{ksi}}{10,000 \text{ksi}} = 73 \times 10^{-6} \text{ in/in} = 73 \mu \epsilon
\]

It is noted that this residual tensile strain is produced before the entire compressive residual stress is attained. The residual compressive strain should be closer to 21.27 ksi (147 MPa) if this correction is included. Hence the computed values are a bit pessimistic, but as a first level approximation this effect has been ignored.

When the specimen is sawed in half across the minimum section, the residual stresses would be permitted to relax. The strain in the central region would be expected to drop to zero and the strain at the root of the radius would be expected to rise to approximately .0022 in/in. This relatively straightforward analytical process yields predicted values for stresses and strains in the loaded, unloaded, and sawn conditions as desired. It is seriously limited by the necessity that an assumption must be made about the depth of the stress concentration effects. It is also limited to simple computation of only the stress and strain in the long direction of the specimen.

**Experimental Issues**

Experimental accomplishment of the testing modeled here has inherent difficulties. Certainly the stress concentration region is small and installation of strain gages in the radius is not trivial for most people. A larger difficulty is that the strains are rapidly changing in the radius, and predicting the strains which will be measured by the gages depends on knowing precisely where the gages are located, and on use of the finite element computations. It may be beneficial to change to a polar coordinate system in the radius to determine the strains attributable to the gages. Additionally, the strain gages are at least somewhat affected by strains under the entire backing material of the gage, not only the strain present under the gage grid. When measuring the location of the gages and predicting the strain that will be measured during the testing, some estimate of what portion of the backing to count as being sensitive is necessary. It is certainly desirable to use the smallest gages consistent with the ability of the installer so that the strains measured are less subject to strain gradient. When sawing the specimen, it is critical that the gages not be sawn asunder, thus destroying the measurement.

When performing the loading, unloading, and sawing steps, it is desirable to have a strain indicator connected to each strain gage during the entire process. This is somewhat cumbersome but reduces the probability of the zero setting being changed during the testing cycle. It is also essential that strain gages be placed at the root of both radii, and on both surfaces of the middle of the specimen. Testing machine jaws do not always grip a specimen evenly from side to side, and if there is any uneven gripping there will be uneven loading applied to the specimen causing bending. The bending strains will add to the tensile strains, most likely changing the values enough to confuse the data significantly. The strain readings should be averaged, either numerically or by wiring as half-bridge, between the oppositely positioned gages to get a better idea of the purely tensile strains.

The entire demonstration could be improved by using a larger specimen; the gages would be easier to install and the gradient under the gages would be reduced. The specimen proposed here is not small, and machines and grips for even larger specimens may be difficult to locate. It is essential that the specimen be sized for whatever system is available. The second author has successfully used this demonstration with a specimen half the size of the one proposed here. The experimental data showed residual strains and relief of those strains on sawing, but the difference from side to side of the specimen were of magnitude similar to the residual strains. Installation of gages in a radius half the size of the one proposed here was too time consuming to be desirable for future demonstrations; hence the proposed size used here.
Analysis

The finite element model for this analysis was constructed as a two-dimensional approximation, assuming no twisting can occur on the actual part. The higher-order quadrilateral element, PLANE82, was used with its plane stress with user-defined thickness behavior option. The geometry and loading shown in Figure 3 is appropriate for a one-quarter symmetry finite element model, Figure 4.

Figure 3. Geometry of Tensile Specimen
Figure 4. One-Quarter Symmetry Model and Element Mesh

The part thickness of 0.5 inches (12.7 mm) is specified as a Real Constant for this model. The material properties were defined by the elastic modulus, $10.0 \times 10^6$ psi (73,000 MPa) and Poisson’s ratio, 0.33. The Bilinear, kinematic hardening stress-strain material model was used to include the yielding behavior for this material. For the nonlinear material behavior, a yield strength of 40,000 psi (275.8 MPa); and a tangent modulus of 0 were defined. This material model assumed an elastic-perfectly plastic stress-strain response.

Four load steps were defined to simulate the experimental sequence. In load step 1, the model was gradually loaded to the final, maximum load of 24,000 lbf (106.76 kN). The initial loads and constraints are shown in Figure 5, with two planes of symmetry and the final load applied as a distributed load on the top edge of the model. In load step 2, the unloading step, the applied load was gradually reduced from the maximum value to zero. Load step 3 was used to delete the applied load and add a vertical (UY) constraint on the top at the centerline of the model which held the part at its final, equilibrium location after the load and unload solution steps. Finally, load step 4 simulated the saw-cut, stress relief through the notches on the part. Figure 6 shows the stress relief, final load step boundary conditions. This figure illustrates that the distributed load has been removed, a vertical (UY) constraint has been added, and the constraints along the bottom edge have been removed to simulate the stress relief cut.
Figure 5. Load Step 1: Loads and Constraints

Figure 6. Final Load Step: Constraints
Analysis Results & Discussion

The simulation results are presented as contour plots of the total y-direction strain (EPTOY) which facilitates comparison to the classical stress analysis techniques developed earlier. Figure 7 shows the model’s response at the end of load step 1 with the full load applied. For the material properties used in this simulation, yielding begins at a strain of 0.004. The contours in the figure show that the material near the notch has experienced yielding at this load condition.

Figure 7. Total Y-direction Strain, Under Full Load

Figure 8 shows the model’s response at the end of load step 2, when the applied load has been removed. Residual strains are observed near the notch.
Figure 8. Total Y-direction Strain, After Unloading

Figure 9 shows the result at the end of load step 4, after releasing the vertical constraint to approximate the saw-cut for stress relief.

Figure 9. Total Y-direction Strain, After Cutting
To look at the process graphically, small regions of the model were isolated and results were extracted to create stress-strain plots for the simulation time history response. Figure 10 exhibits a “patch” of six elements near the center of the model that approximate the size and location of one strain gage (named gage “A”). Similarly, Figure 11 shows a group of nodes on the edge of the notch that represent the size and location of another strain gage, gage “B.”
At location gage “A,” the element centroid results for y-direction stress and total y-direction strain were extracted and averaged for the six elements used to approximate gage “A.” Figure 12 show the stress-strain behavior for gage “A” over the entire simulation. Figure 13 shows just a close-up portion of the same stress-strain data.

![Figure 12. Stress-Strain Response at Gage "A"](image)

At location gage “B,” nodal results for y-direction stress and total y-direction strain were extracted and averaged for the nodes used to represent strain gage “B,” in the notch. Figure 14 show the stress-strain curve in the notch for the entire simulation. Figure 15 shows just a close-up portion of the same stress-strain data, near the curve’s origin.

![Figure 13. Close up, Stress-Strain Response at Gage "A" (Showing the Stress Relief Cut Response)](image)
Conclusion

The agreement between the classical and finite element analyses is fair but not as similar as had been both hoped and expected. With the full load applied, the strain at gauge A and B are predicted to be .003 and .0062 by classical methods and .0021 and .0075 by FEA respectively. When the load is removed, the predicted strains are both approximately .000073 by classical methods and .00002 and .0018 by FEA. When the specimen is saw cut, classical methods predict strains of zero and .0022 at gauges A and B respectively, and FEA predicts zero and .0035 at A and B. Additionally, it was hoped that a relatively large area of moderately uniform residual stress could be reliably produced for calibration of residual stress measurement methods, and the hoped-for large area is not evident.
Even though not identical results are predicted by the two analyses, the proposed specimen and testing would reliably produce residual stresses large enough to measure reliably. As a demonstration of one residual stress cause and its measurement, the procedure proposed here is quite useful.

Finally, the difference between the classical and finite element strains on loading indicate that the stress concentration charts for a tensile member with notches in the sides may not apply to this situation where the center of the notch radius is not exactly aligned with the edge of the specimen. Many stress concentration charts were developed by brittle models and photoelastic methods, both of which are entirely linear when developing the charts, and may not be representative of action after initiation of yielding. Designers using classical methods must be very careful in using published data to be certain that they are using it exactly as developed; FEA has the advantage of analyzing the geometry of the situation at hand.

**References**

