Designing and evaluating tooling used to form a hydraulic end connection on tubing using MSC.Marc

Philip Van Riper
Senior Analysis Engineer
Eaton Corporation
Technical Center
1660 Indian Wood Circle
Maumee, Ohio 43537
Phone (419) 891-7806
E-mail phillipcvanriper@eaton.com

Abstract

This paper uses the finite element method (FEM) to virtually form a hydraulic end connection on carbon steel tubing. The benefits of this approach are twofold. One, the design of the end connection itself can be evaluated to determine if unacceptable levels of strain are introduced during the forming operation. This can be due to the wrong material being specified or dimensional constraints that do not allow the material to flow sufficiently. Second, the tooling design and set-up can be evaluated to determine their capability to form the end connection. Design changes and optimizations are made prior to producing prototypes.

Keywords: Virtual manufacturing, plasticity, residual strains, MSC.Marc, metal forming, FEM

Introduction

The design of tooling used for forming hydraulic end connections on tubing is usually done by generating a design based on the end connection desired and prior experience. The tooling is made and then tested. The initial tooling design is done by skilled tool designers with a large amount of experience in the tube forming industry. Even with this experience, it is often necessary to perform several forming trials because it is not always evident a priori how the material will react to the tooling geometry or if any crack initiation sites are created.

Based on the results of the forming trial, changes to the tooling, process or material selection are made and the forming repeated. Once the dimensional parameters are met, the formed hydraulic end connection is performance tested. If the testing reveals flaws in the design, the hydraulic end connection would need to be redesigned and then the tooling would have to be redesigned and more prototypes made. This process is costly in terms of both money and time.

The objective of this paper is to demonstrate how the use of the FEM early in the design process can eliminate design iterations, show problems with the proposed tube design and provide valuable insight into the strain imposed on the tube at different stages of the forming operation. The evaluation of different materials and material conditions is very easy once the forming operation is defined in the analysis.

This particular end connection is made in five different sizes. The first size was developed without the aid of FEM. This added greatly to the cost and development time of the project. All subsequent sizes utilized the methods outlined in this paper.

Definition of Terms

ε_f	Fracture Strain
A_r	Reduction of Area
σ	True Stress
K	Strength Coefficient
n	Strain Hardening Coefficient
ε	True Strain

Problem Definition

The connection assembly detail is shown in Figure 1. The connection consists of a male portion (the part being examined in this paper), a female portion, housing the fluid seal and the latching ring, and release sleeve. The connection is made when the male portion is slid into the female portion. The ramp on the male portion forces the latch ring over a rib which then locks the connection together. To release the connection the ring is forced back over the male rib by the release sleeve being forced forward by insertion of the release tool between the release sleeve and a large machine diameter used to provide support for the release tool.

The male end of the connection of Figure 1 is currently machined out of bar stock, shown in Figure 6, and then brazed onto tubing. To eliminate the leak path of the braze joint it is proposed that the end connection be formed directly upon the tubing as shown in Figure 7.

It is recognized that the exact representation of the Figure 6 design cannot reasonably be formed onto the tube. The small radii and tight tolerances called out in the machine design do not account for the flow of the material necessary to form the end connection. Therefore, the critical areas of the male connection are identified and the rest of the dimensions and features are changed to allow for material flow and so that large stress concentrations are not imparted onto the tube.

In the initial formed design, the release sleeve support diameter was designed to have the same diameter of the machined design. It was thought that the width should be similar to standard tube bumps. This resulted in the bead having a very tight bend radius however the bead design is similar to other tube beading designs shown in Reference 6 and it was felt that this would be adequate for the current design.

As stated earlier, the first size of the end connection was developed using the older method of trial and error. The sample was formed to the required dimensions, but failed in performance testing prior to meeting the minimum test requirements. A material analysis showed that during the forming operation the release tool support diameter experienced crack initiation and propagation. There were no visible failures during the forming operation. A micrograph of this region is shown in Figure 2. It is clear that the crack is due to the tight bend radius of the bead. The bead designs used in Reference 6, among others, are supported on both sides during installation, whereas in this design the bead is supported on neither side and is subjected to bending stresses during operational loading.

Analysis

It is clear that the tubing experiences large plastic strain during forming. To more accurately model the plasticity, the specified tubing is subjected to tensile testing per Reference 2. In this test, a length of the tubing is used as the tensile specimen. To keep the tube from collapsing in the crimp jaws of the tensile machine, two tapered plug are inserted into the ends. A total of five samples were tested.

The tensile data includes the stress-strain curve from initiation of loading until the ultimate load is reached. This gives a good example of the plastic strain versus stress. Additionally, the fracture strain is calculated by equation (1) based on the reduction of area. The fracture strain is used to determine how close the tube is to fracture during the forming operation.

$$\varepsilon_f = \ln \frac{100}{100 - A_r} \tag{1}$$

The power law is used to represent the plasticity with the strain hardening coefficient and strength coefficient determined from the tensile data of five samples. The strain hardening coefficient and strength coefficient are determined from a least squares fit of the log-log data of the true strain and true stress.

$$\sigma = k\varepsilon^n \tag{2}$$

The material data used in this analysis is given in Table 1.

The geometric design of the forming tooling, not its strain response during the forming operations, is of importance in this paper. The decision to use tool steel for the forming tooling is already made. Since the tooling is so much stiffer than the tubing, the tooling is represented by rigid bodies.

An important variable that influences the final shape, strain profile and tooling designs is the amount of tubing that extends past the clamping area and interacts with the forming tooling. This is known as the forming length of the end connection. It is desirable to make this length as short as possible to minimize the amount of tubing required.

The design and forming of the tubing as shown in Figure 7 is axysymmetric about the longitudinal axis of the tubing. A 2D Analysis is done using element 10, invoking the updated Lagrangian and multiplicative large strain plasticity options.

There are several different tools and tool sets acting in sequence to form the end profile on the tubing. The virtual representations of these tools are shown in Figure 8. The movements of each tool and required set of tools is controlled by position through tables and by using the approach option. It is not known a priori the intermediate position of the tubing during various forming operations so the tooling is positioned at a distance away from the part and in one step moved to where it just touches the tubing using the approach option. This saves valuable computing time.

The model is evaluated to determine if the desired shape is achieved and if the strains in the end connection are large enough to cause fracture. This evaluation is done after the application of each set of tools prior to the tool being released. This is important because the maximum strain in the part at each forming step occurs with the full application of the tooling, before the part has a chance to stress relieve. If either parameter is unacceptable, adjustments are made to either the tooling or the form length of the tubing.

Discussion

The first consideration is to verify the FEA model and modeling assumptions by bounding the solution with known classical solutions or comparing to test results. Since the first size of the end, connection was formed and tested prior to utilizing FEA the latter approach was chosen.

Figure 2 is a micrograph of the release sleeve bead cross-section prior to testing. A crack has initiated and propagated at the base of the bead but has not broken through to the tube surface. It is very difficult to detect this type of defect during the formation of the tube end connection. The defect does not become apparent until the end connection is subjected to performance testing, which is what happened to the first size design.

The virtual representation of the area is shown in Figure 3. The FEA results show that area of the tube with the highest strain is at the internal surface of the release tool support bead. The strain of 1.646 is above the fracture strain and thus indicates that crack initiation at this location is likely.

The FEA results are slightly different from the micrograph in that the two sides of the bead do not touch. This discrepancy could be due to the fact that there is no separation of the elements due to the crack initiation and propagation making the area stiffer in the simulation than the physical model. Recall that the plastic behavior of the material model is based on a least squares fit of data taken from five different tensile tests, whereas the exact material properties of the physical model are not know. It is very difficult, using generalized macro properties, to accurately model the deformed geometry of a localized phenomenon such as crack initiation. The addition of more elements at this location could have provided a more accurate representation of the deformation.

The dimensions of the physical model were compared to the results from the simulation and showed excellent agreement. The FEA model was able to accurately predict the final dimensions of the part, the location of largest strain and that the strain at this location could cause fracture during forming. Its only difficulty was in the internal dimensions of the release sleeve support bead, which has been previously discussed. It is therefore concluded that the FEA model is verified by the physical results and it was decided to use the same modeling and material assumptions for the remaining sizes.

With the modeling and material assumptions verified, the problem of redesigning the release tool support bead began. It was decided to elongate and shorted the height of the bead to reduce the severity of the corner on the inside surface. The only purpose of the bead is to provide leverage for the release tool so that the release sleeve is pushed forward and the connection disassembled. Several designs were evaluated prior to finding the best alternative in terms of providing enough leverage and a low amount of strain. This design is shown in Figure 4. The location of the largest strain in this area is split between the inner surface and at the beginning of the bead. The magnitude of the strain has been cut in half to .805, which provides a factor of safety of 2 with respect to crack initiation.

With this redesign, as often happens, the release tool support bead in no longer the location of the largest strain in the formed tube but has now moved to the inner surface of the male rib as shown in Figure 5. The strain at this location of .945 is 70% below the strain at the release tool support bead of the original design. Thus, this design is superior in terms of strain when compared to the original design. The robustness of this design was shown during the subsequent performance testing of the formed tube and incorporated into all subsequent sizes.

The FEA of the other sizes proceed in a logical fashion with numerous adjustments to the tooling and tube form length.

Conclusion

The first design, which was done using the old method of trial and error, took 14 months and five tool revisions, and still did not pass performance testing. It was then decided to use the FEM. The FEM was able to detect areas of potential failure that could not be easily found in the physical model and give a complete strain history during the entire forming operation. The FEA modeling assumptions were verified by comparing the numerical dimensional and strain results to the physical results of the formed end connection done prior to utilizing FEA. The assumptions used in this model were used to redesign the release tool support bead and in subsequent analyses of additional sizes.

Using the FEM, the tooling and tube end connection were designed and verified in two weeks and proved out during forming and testing for each size. The total time required to analyze, make prototype tooling, form the end connection and finally subject the end form to performance testing was 2 months per each additional size, with a lot of the work being done in parallel. The performance testing of the physical parts matched the simulation results and none needed to have another design iteration.

The analysis procedure described in this paper does not replace but rather enhances the ability of the tooling designer. Instead of waiting months to make prototypes and test them, the tooling designer can real results in a matter of a few hours. These results are very detailed and allow a deep understanding of the process and its effect on the tooling and the formed part. Early use in the design process of the FEM enables the design to increase the robustness of the tooling and tube form while at the same time saving development time.

Acknowledgments

Steve Hauber, Chris Schadewald and Todd Vogel

References

1. Shigley, J.E., and Mischke, C.R., *Standard Handbook of Machine Design*, McGraw-Hill 1986.
2. ASTM Specification, E8-98, *Standard Test Methods for Tension Testing of Metallic Materials*.
3. *MSC.Marc 2003 User's Manual Vol. A*, MSC.Software Corporation, Los Angeles, CA, 2003.
4. Metals Handbook, Volume 1 10th Edition, 1990.
5. Cook, R. D., Malkus, D. S., Plesha, M.E., *Concepts and Applications of Finite Element Analysis*, Third Edition, John Wiley and Sons, 1989.
6. SAE J1453
7. Hertzberg, R.W., *Deformation and Fracture Mechanics of Engineering Materials*, Third Edition, John Wiley and Sons, 1989
8. Ellyin, F., *Fatigue Damage, Crack Growth and Life Prediction*, Chapman & Hall, 1997.

Figures and Plots

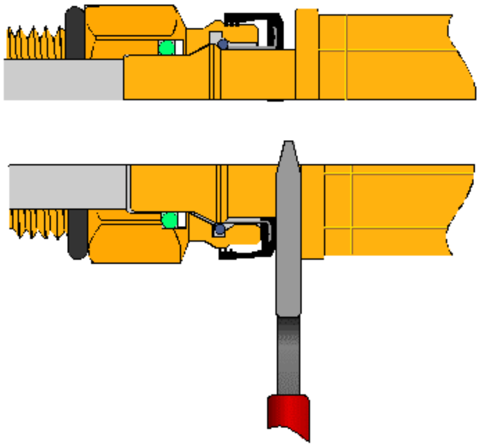

Figure 1 Connection Assembly Detail

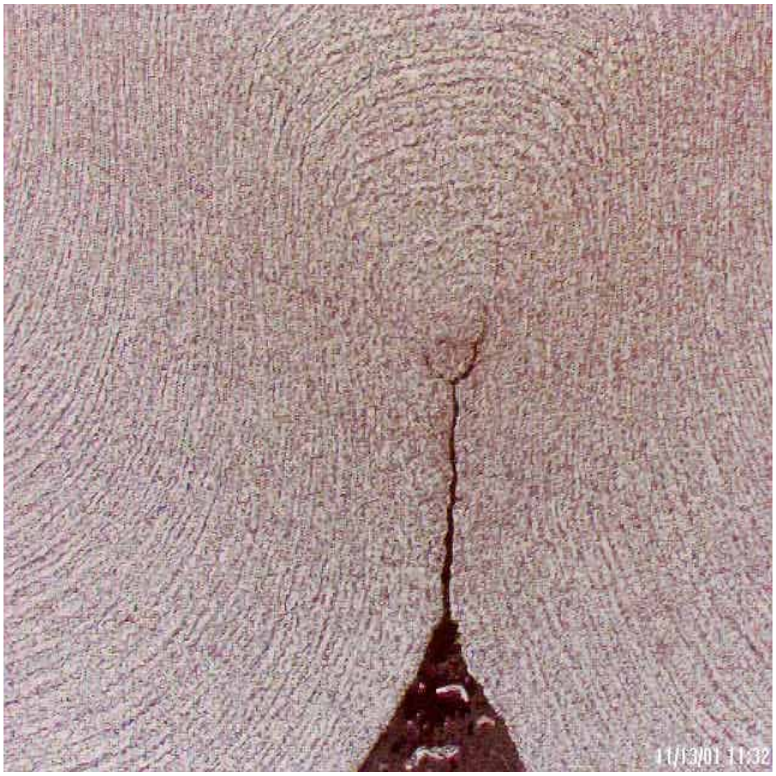

Figure 2 Release tool support diameter micrograph

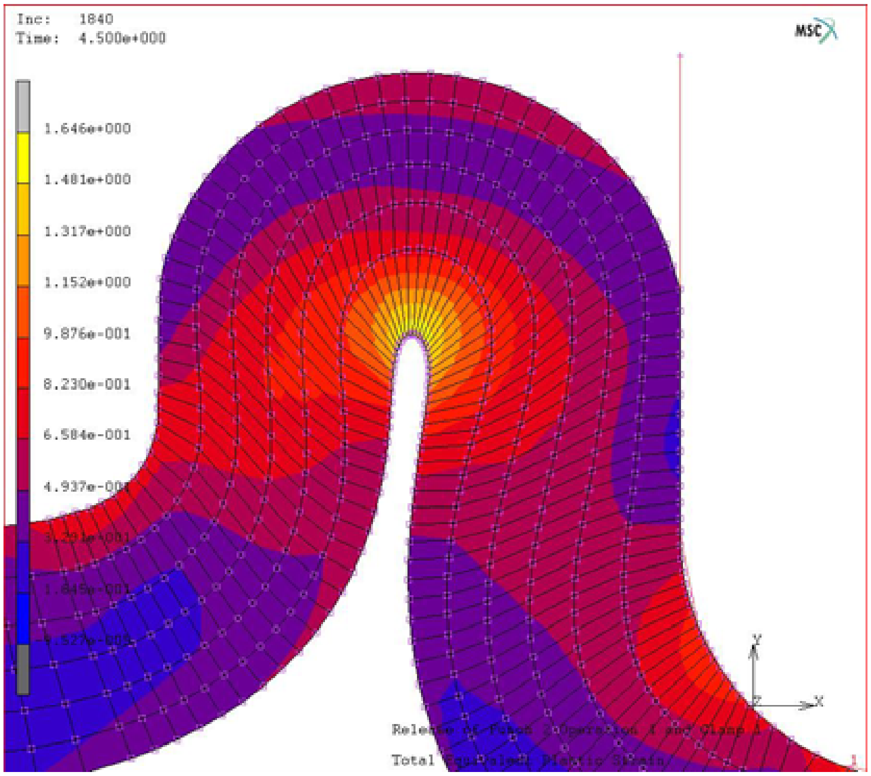

Figure 3 Support Bead FEA results

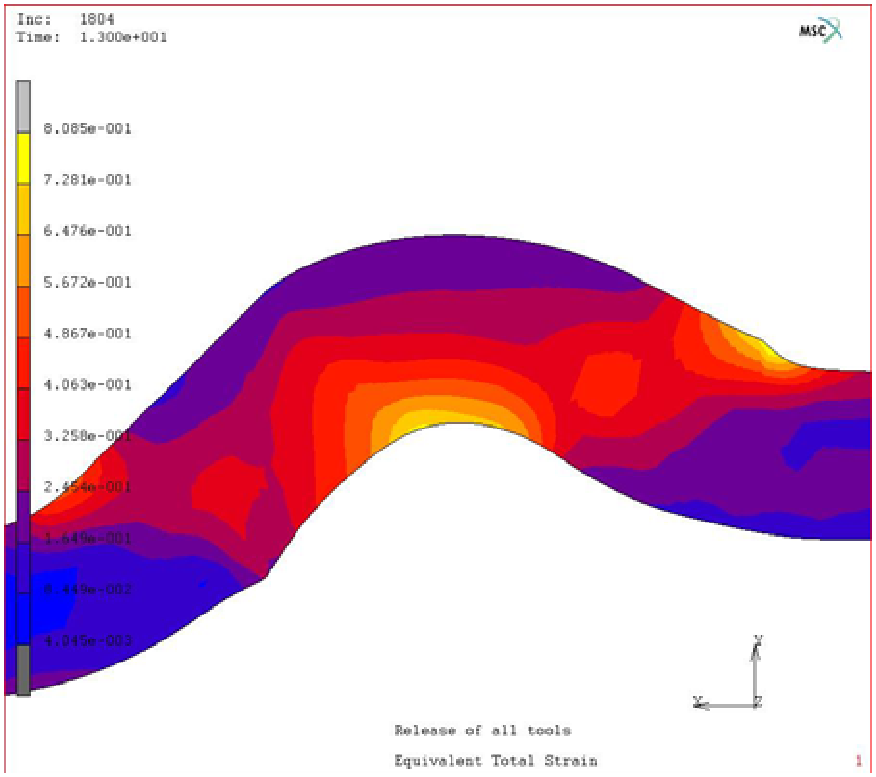

Figure 4 Strain Results for new release tool support bead

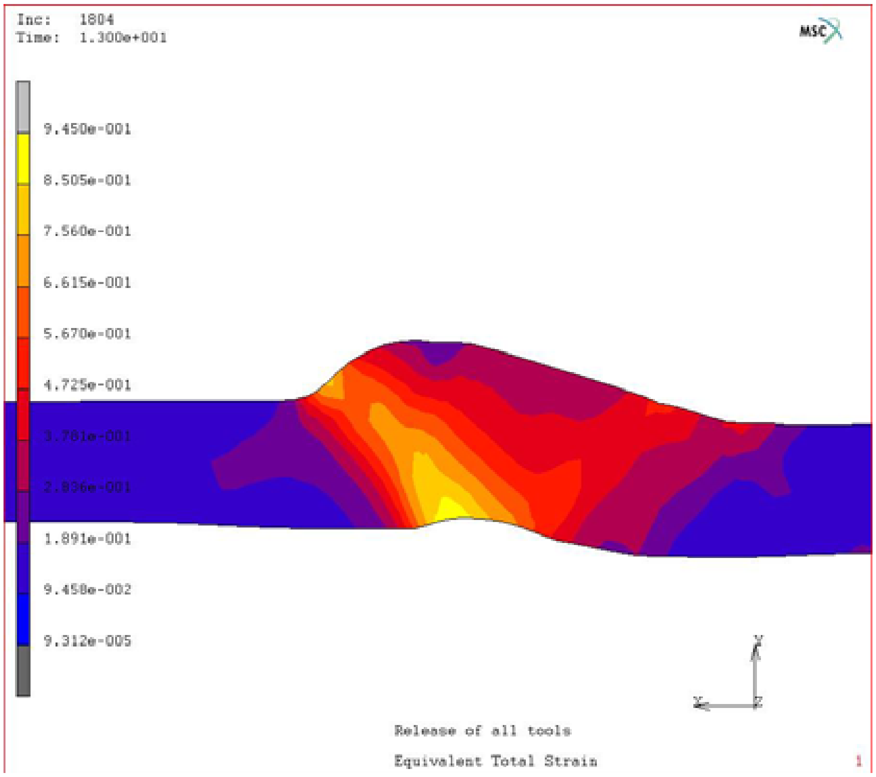

Figure 5 largest strains in end connection with redesigned release tool support bead

Table 1 Material Data

Sut	50 ksi
Sy	30 ksi
k	173 ksi
n	.268
RA	.80
εf	1.63

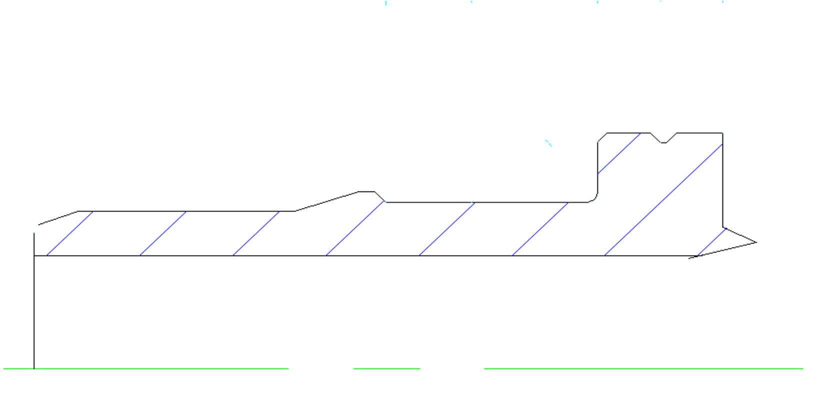

Figure 6 Machined Male STC

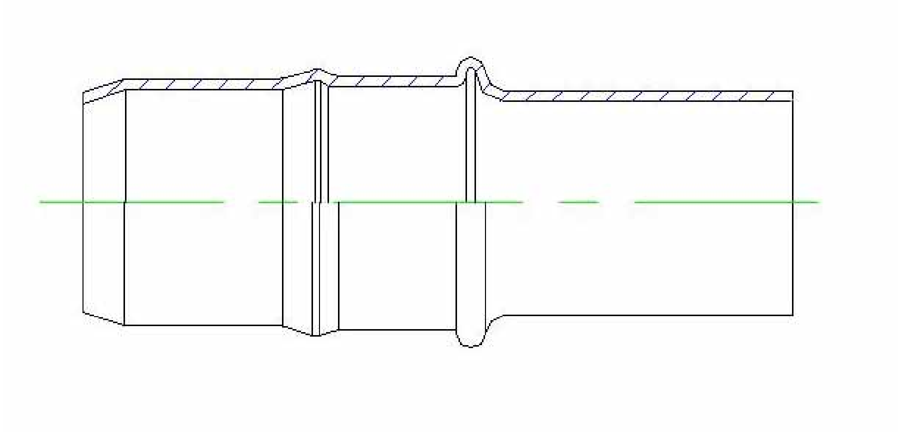

Figure 7 Formed Male STC

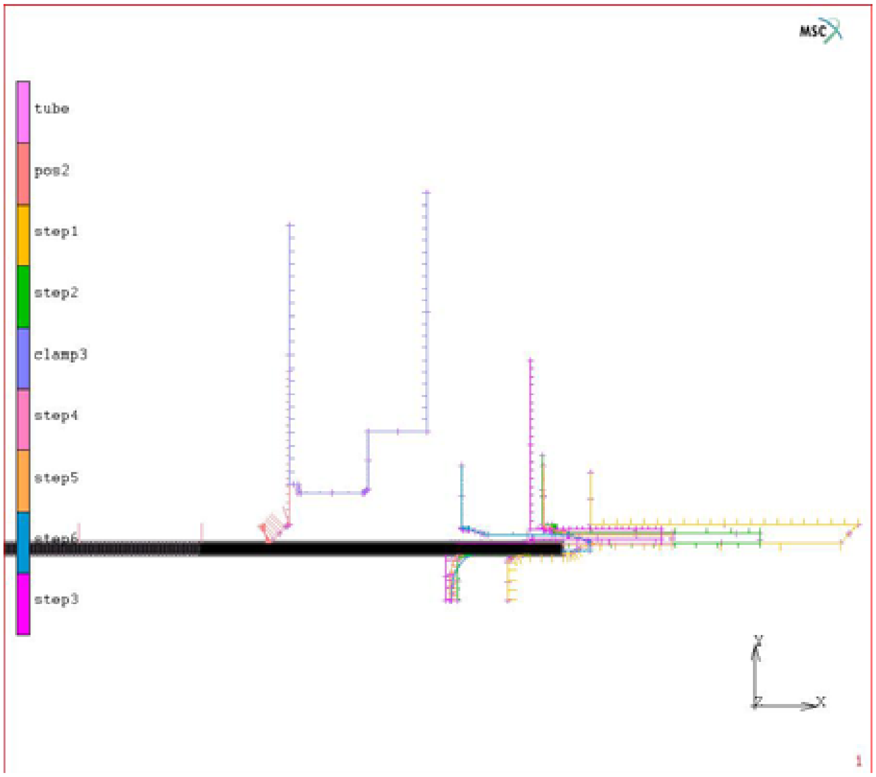

Figure 8 MSC.Marc FE Model showing all contact bodies

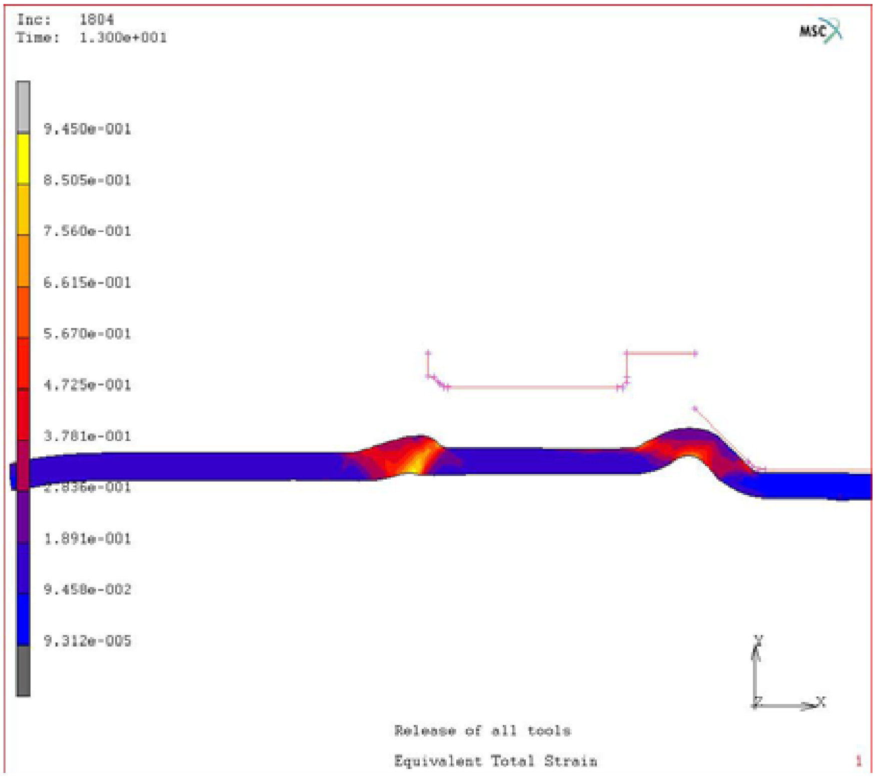

Figure 9 Final Design after all tooling has been removed