Structural Analysis of Welded Connections Using Creo Simulate™

Jonathan Polom
Design Engineer, U.S. Army TARDEC
RDTA-SIE-CSI-ES-MDT
jonathan.m.polom.civ@mail.mil

UNCLASSIFIED: Distribution Statement A. Approved for public release.

Report Documentation Page		Form Approved OMB No. 0704-0188

1. REPORT DATE **07 APR 2014**	2. REPORT TYPE **Academic Report**	3. DATES COVERED **02-01-2014 to 25-03-2014**
4. TITLE AND SUBTITLE **Structural Analysis of Welded Structural Analysis of Welded**		5a. CONTRACT NUMBER
		5b. GRANT NUMBER
		5c. PROGRAM ELEMENT NUMBER
6. AUTHOR(S) **Jonathan Polom**		5d. PROJECT NUMBER
		5e. TASK NUMBER
		5f. WORK UNIT NUMBER
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) **U.S. Army TARDEC,6501 East Eleven Mile Rd,Warren,Mi,48397-5000**		8. PERFORMING ORGANIZATION REPORT NUMBER **#24632**
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) **U.S. Army TARDEC, 6501 East Eleven Mile Rd, Warren, Mi, 48397-5000**		10. SPONSOR/MONITOR'S ACRONYM(S) **TARDEC**
		11. SPONSOR/MONITOR'S REPORT NUMBER(S) **#24632**

12. DISTRIBUTION/AVAILABILITY STATEMENT **Approved for public release; distribution unlimited**

13. SUPPLEMENTARY NOTES

14. ABSTRACT

Creo Simulate (previously marketed as Pro/Mechanica) finite element analysis (FEA) software is part of the Creo mechanical CAD software suite. Design engineers using the Creo Parametric modeling environment often use this tool to quickly analyze the mechanical performance of parts and assemblies, including welded structures. The rapid exchange of geometry data between the design and analysis tools can substantially reduce the amount of time necessary for an engineer to complete an analysis when compared to other commercially available products. This project will investigate the structural analysis capabilities of Creo Simulate for weldments by examining several welded structure scenarios using three-dimensional solid finite elements: a welded plate beam, tubular support member and sheet metal frame. The methods documented and used to create and interrogate these models can serve as the basis of recommended procedures for weldment analysis using this analytical tool.

15. SUBJECT TERMS					

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT **unclassified**	b. ABSTRACT **unclassified**	c. THIS PAGE **unclassified**	**Public Release**	**13**	

Abstract

Creo Simulate™ (previously marketed as Pro/Mechanica) finite element analysis (FEA) software is part of the Creo® mechanical CAD software suite. Design engineers using the Creo Parametric™ modeling environment often use this tool to quickly analyze the mechanical performance of parts and assemblies, including welded structures. The rapid exchange of geometry data between the design and analysis tools can substantially reduce the amount of time necessary for an engineer to complete an analysis when compared to other commercially available products. This project will investigate the structural analysis capabilities of Creo Simulate™ for weldments by examining several welded structure scenarios using three-dimensional solid finite elements: a welded plate beam, tubular support member and sheet metal frame. The methods documented and used to create and interrogate these models can serve as the basis of recommended procedures for weldment analysis using this analytical tool.

Introduction

Creo Simulate™ is a finite element analysis (FEA) tool and part of the Creo® mechanical CAD software suite. Creo Parametric™ is the design portion of the Creo suite with capabilities for creating solid part geometries and multi-level assemblies of parts. A module system enables additional features to be added to the design capabilities of Creo Parametric, such as rigid body dynamics simulation and finite element analysis. Creo Simulate is accessed as an extension module in the design program.

The close integration of Creo Simulate with the design environment often makes it an attractive choice for analysis. Geometry information from parts and assemblies automatically transfers to the FEA tool which eliminates any need to export geometry from a CAD tool and import it into a separate FEA program. Additionally, the software uses polynomial finite elements so a coarser mesh may be used without negatively affecting solution accuracy. Geometry meshing is mostly an automated, hands-off process for the analyst. Hard curves, points and element size limits can be used to refine a mesh, but are only required in special circumstances.

Using Creo Simulate to assess the mechanical performance of welded structures presents specific difficulties not encountered when simulating solid parts (billet, castings, and forgings) or bolted assemblies. This report will highlight approaches for analyzing primary and secondary welds on built-up plate beams, tubular supports and sheet metal frames. Models documented in this report employ one of two fundamental strategies using three-dimensional solid finite elements: modeling the welded joint with solid fillers and contacts or a stress measurement method. A discussion of benefits and drawbacks associated with each of these strategies will be included.

Welded Plate Tee

This section details assessing web-to-flange weld adequacy on a steel tee section. Problems 1 through 3 in section 2.6 of *Design of Weldments* (Blodgett) solve this problem using Bernoulli beam equations. The section construction consists of a flange and web plate joined with continuous fillet welds, shown in Figure 1. Loading and boundary conditions are depicted in Figure 2. Length units are inches. Material properties for steel were assigned to all parts in the analysis: Poisson's ratio of 0.27 and Young's modulus of 29,000,000 psi. However, in the context of a theoretical (Bernoulli) beam, these material properties are only needed to estimate beam deflection. Bending moments and stresses only depend on the geometry, boundary conditions and loads.
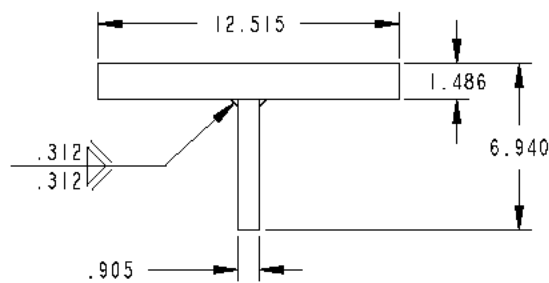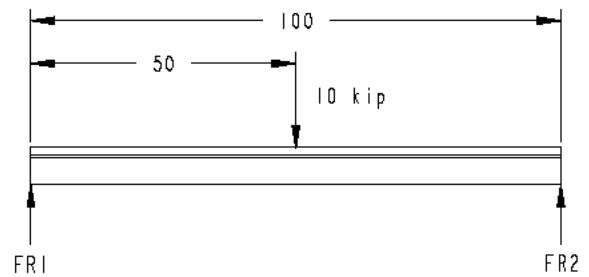
Figure 1: Tee Section

Figure 2: Loading and Boundary Conditions

The plate beam is modeled as an assembly using four parts: a flange plate, web plate and two fillers representing the deposited weld metal. A non-linear contact interface (highlighted red, Figure 3) is placed at the contacting upper web surface and lower flange surface. The weld filler is automatically bonded to the web and flange parts along its length. All degrees of freedom are constrained for the lowest edge of the web on the first beam end (blue arrows, Figure 3), and translation is allowed along the longitudinal beam direction at the opposing beam end (lower blue arrows, Figure 4). Failure to constrain these degrees of freedom will prevent the solver from running the model. The 10 kip load is shown in Figure 5 which is applied to a curve between two points on the top of the flange. A "Hard Curve" AutoGEM mesh control integrates the curve into the meshed geometry. These constraints realistically approximate the physical geometry of a point loaded, simply supported welded plate tee without inadvertently increasing stiffness of the member.
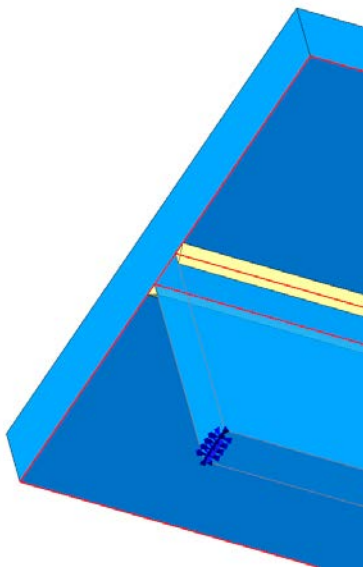

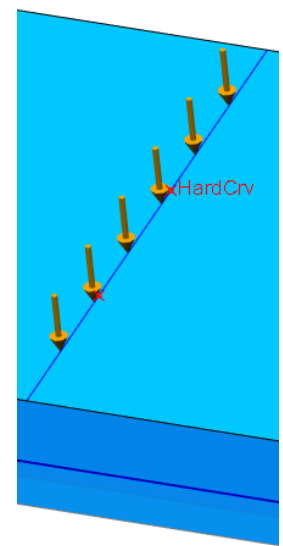**Figure 3: Beam End 1**

Figure 4: Beam End 2

Figure 5: Load Application

Use of a contact interface requires using the non-linear solver and load histories. The software automatically creates two load steps when this option is selected, with the second load step containing results with contact effects. Adding additional load steps is possible and may

increase solution accuracy, but was not necessary for this analysis. The single-pass adaptive solver is the only available solver option for non-linear analyses.

Model results for the beam agree well with manual calculations shown in section 2.6 of *Design of Weldments* (verified). The maximum longitudinal stress in the FEA model is 21,646 psi on the bottom of the web, within 1% of the textbook value. Shear stresses are negligible at the bottom of the web in this beam. Figure 6 shows the von Mises stress across the beam, and a stress of no more than 4,500 psi in the fillet welds. Horizontal YZ-plane shear stress also agrees well with textbook calculations near the fillet welds and some representative point value stresses are shown in Figure 7. Some localized stress concentrations occur along the lower weld toes and weld root (orange in figure). The mesh is visible in both figures.

Summary

A finite element model using 3D solids in Creo simulate may be a very efficient option for assessing the adequacy of secondary welds on a fabricated plate structural section. Weldment assemblies generally don't include solids for weld filler, so some time may be required to create the respective filler geometries for the analyzed joints. Use of as-is part geometry leads to minimal modeling overhead for this approach, with minor modifications (usually just de-featuring) if any. As the model grows in size, this approach may become impractical due to the large demand on computational resources. This model required an elapsed time of 84 seconds to solve, indicating developing a symmetrical model wouldn't have reduced total modeling time. Larger models may benefit from the exploitation of symmetry or shell simplifications.
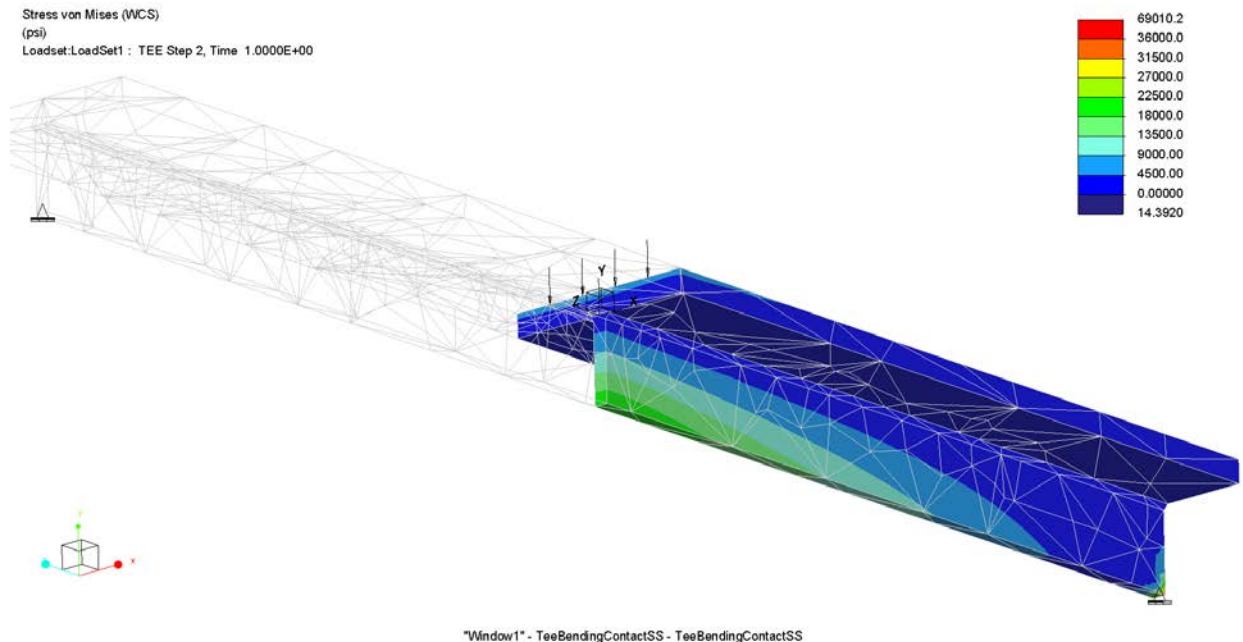
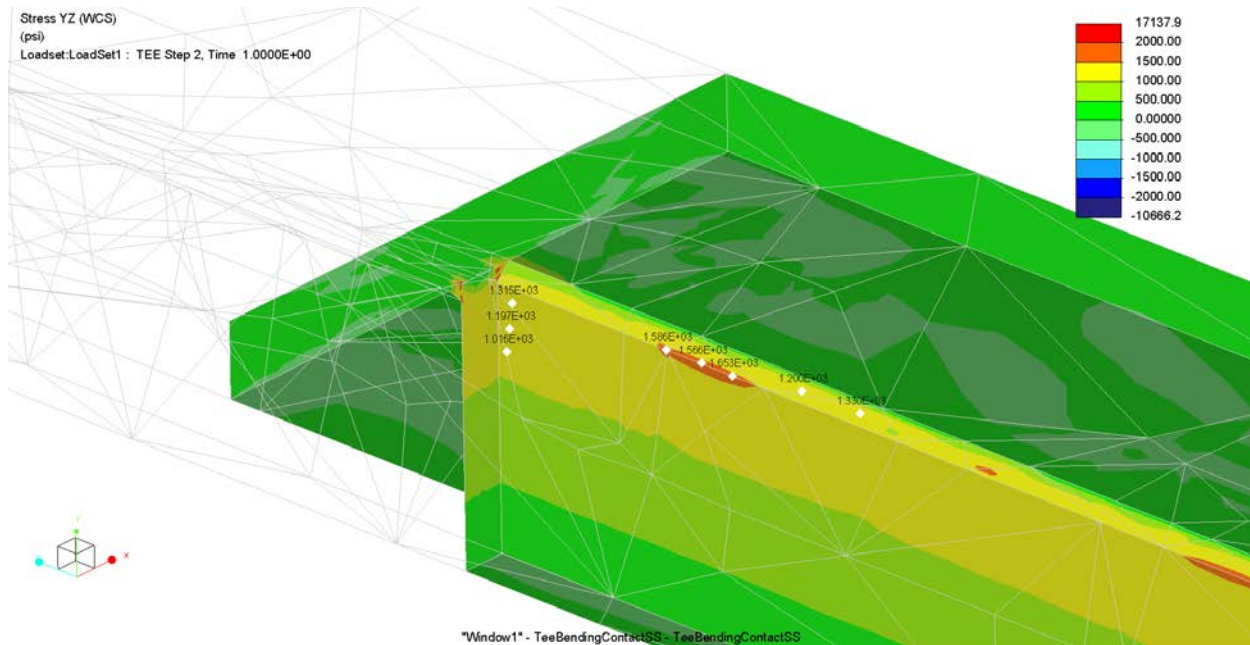

Figure 6: Mises Stress for Beam

Figure 7: Horizontal Beam Shear

Tubular Support Arm

This section assesses primary fillet and v-groove welds in an arm support constructed from 1.50 inch O.D., 0.125 inch wall ASTM A513 DOM steel tube. The arm needs to resist a 127.75 lb. vertical load at the end of the upper tube without yielding. Two sections of mandrel bent tube are joined with CJP groove welds and attached via two slightly unequal leg fillet welds to a base flange. The end of the top tube has a welded-in cap that provides female screw threads for an attachment. An internal coupler provides backing and alignment for the groove welds.

As with the previous model, material properties of steel were assigned to all parts in the assembly. A value of 0.27 for Poisson's ratio was used, along with a Young's modulus of 29,000,000 psi. Figure 8 depicts the loading and boundary conditions used for the analysis. The groove and fillet welds analyzed in this study are depicted in Figure 9, highlighted in yellow.
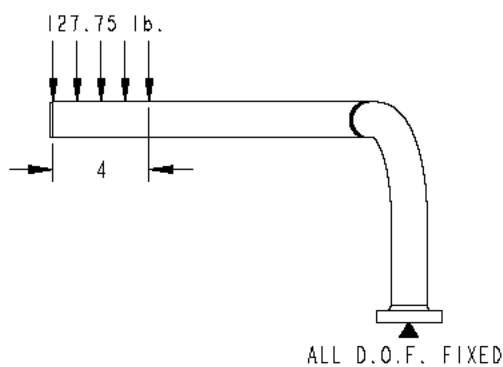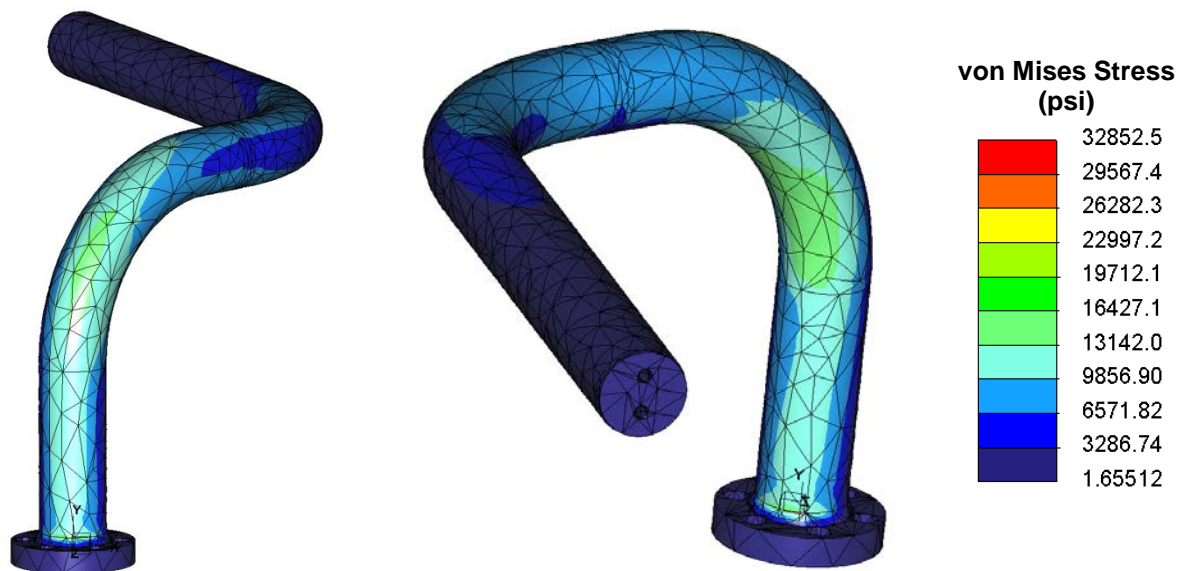

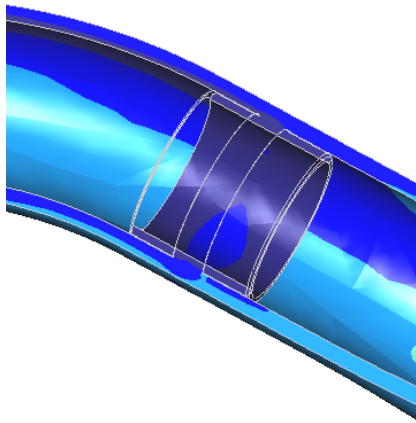**Figure 8: Loading and Boundary Conditions**
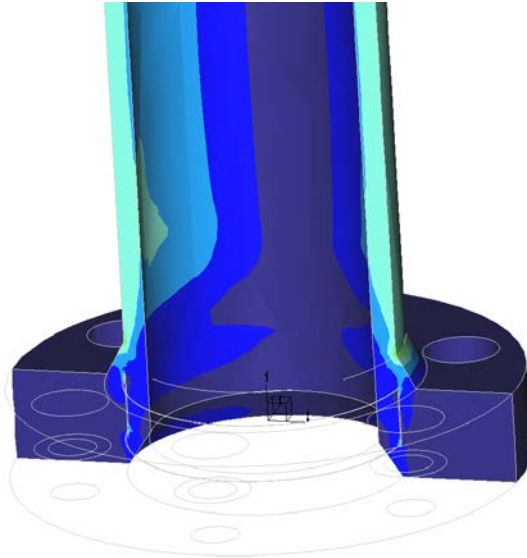

Figure 9: Tube Welds

This model did not require use of non-linear contact interfaces due to the use of CJP groove welds and clearances designed into the parts. The default bonded interface was used for this analysis, and the model was solved with the linear single-pass adaptive solver. A volume region 4 inches deep defined at the end of the upper tube allowed use of a distributed bearing load.

This weldment is subject to the rules of the structural steel welding code (AWS D1.1) which sets allowable stress limits for CJP groove welds and fillet welds. The base material being joined is grade 1020 steel which has an approximate yield point of 42 ksi when annealed (note with grade 1020 steel this value is not guaranteed by the material specification). Due to the precision welding requirements in this part, the preferred welding process is manual GTAW with a filler metal that produces 70 ksi ultimate strength weld metal. Fillers of this classification tensile strength that are compliant with AWS A5.18 have a 58 ksi yield point, resulting in an overmatched joint.

von Mises Stress (psi)

32852.5
29567.4
26282.3
22997.2
19712.1
16427.1
13142.0
9856.90
6571.82
3286.74
1.65512

(close up of CJP groove welds)   (close-up of fillet welds)
Figure 10: von Mises Stress Results

The von Mises stress results (Figure 10) look promising for this assembly, with a global maximum stress of 32.9 ksi, which is less than the yield point of the base material. Stress is concentrated on diametrically opposed surfaces of the vertical portion of the support, oriented in the direction of the load which is creating a bending moment on this section of tube. The fillet welds that attach the tube to the base are the most highly stressed welds and have a maximum stress concentration of 15 ksi. The two v-groove welds shown in Figure 10 (bottom left) have a max von Mises stress concentration of 7 ksi.
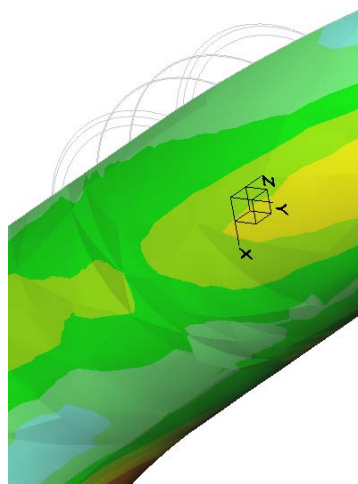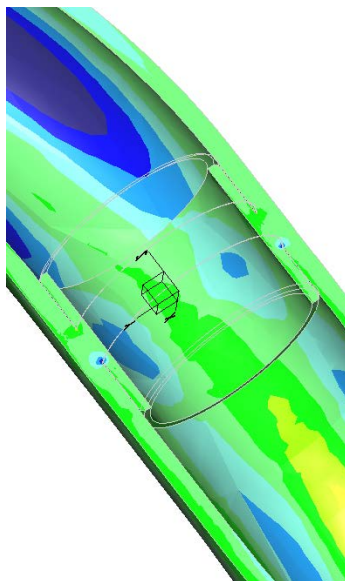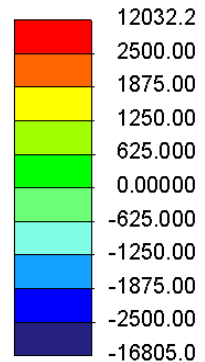

Figure 11: XY Plane Shear Stress

Figure 11 shows a top cross-section and bottom view of the shear stresses in the plane of the two v-groove welds. AWS D1.1 sets the maximum shear stress magnitude for groove welds to 30% of the filler classification tensile strength (21 ksi). There is an additional limitation that the

shear stress on the base metal can't exceed 40% of the material's yield strength (16.8 ksi). Several stress measurements can be taken over the geometry of interest using the *Dynamic Query* tool in Creo Simulate that can be RMS averaged to produce a representative estimate of the nominal stresses in that plane. Nominal stress values are essential when assessing a structure for AWS D1.1 compliance. For this tube weldment, shear stress values in the welded regions were below AWS maximums in all places, so an average was unnecessary. Some localized yielding may be seen in certain circumstances, however, an average estimate of the nominal stress should be less than the AWS allowable stress.

Summary

Finite element modeling excels at calculating stress values for complex three-dimensional loading scenarios, such as this tube support. Ensuring compliance with AWS D1.1 recommendations is a more complex task when using FEA to obtain stress estimates, due to the use of nominal stress limits in the code. An averaging technique using the result query tools allows nominal stress estimates to be calculated from the fine-grained FEA model results. FEA also permits the engineer to examine stress components, such as the von Mises component, that include shear and tensile/compressive stresses.

Aluminum Sheet Metal Shelf

This section describes a faster but less accurate way to extract the information necessary to size secondary fillet welds using measure features in Creo Simulate. A shelf is fabricated out of .125 in. 5052-H32 aluminum plate with brake formed edges and supports. Four diagonal stiffeners made out of the same material are to be welded onto the bottom of the shelf to increase bending stiffness. This study's objective is determining the horizontal shear that the fillet welds need to transmit. Elements of the structural aluminum welding code are followed (AWS D1.2).

The shelf has two loads each of 321.2 lbs. distributed over a rectangular region (Figure 12) on the top of the shelf. The load represents a static approximation of a dynamic load that the shelf must resist without yielding. The shelf is designed to be supported at two flanges with screws; however this was simplified in the analysis by fixing all degrees of freedom of a surface on each flange. All parts in the assembly were designed to contact when placed into the assembly model, which allows use of the default bonded interface.
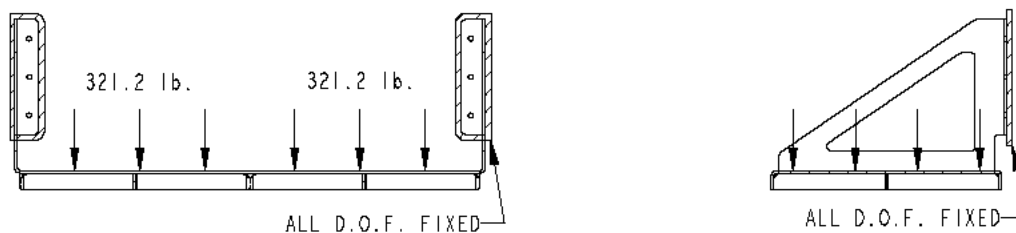

Figure 12: Loads and Boundary Conditions

Young's modulus was set to a value of 10,200,000 psi and the Poisson's ratio was set to 0.33 for all parts in the model.

A simulation measure is defined to measure the maximum horizontal (YZ-plane) shear at the contacting upper surface of the rib and the bottom of the shelf surface. Vector stress components are relative to a particular coordinate system, and since these ribs are installed at an angle relative to the assembly's world coordinate system, a separate coordinate system needs to be defined (Figure 13) with the proper orientation for the desired stresses.
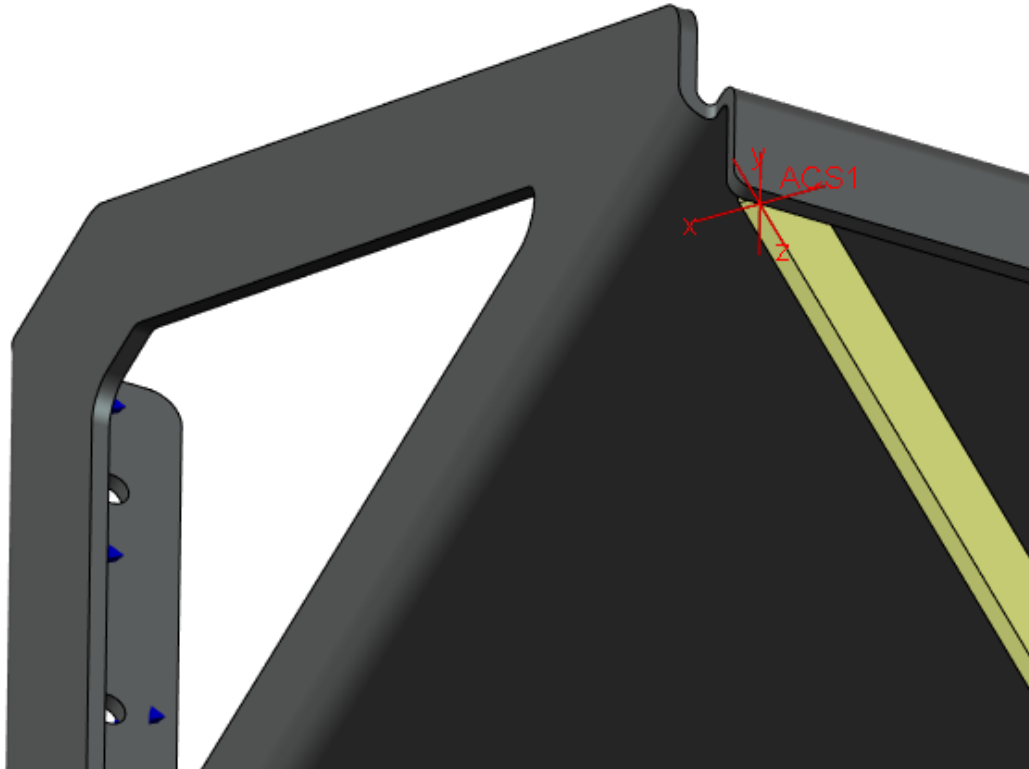

Figure 13: Coordinate System Definition

A measure is created by accessing the "Measures" manager dialog box. The measurement is configured as shown in Figure 14. Measures in have limited functionality, and it's not possible for Creo Simulate to average a measure over a region. In this case, the maximum value of the measure over a selected geometric region (a surface in this case) is recorded. The surface searched over for a maximum is shown in Figure 15 with a blue outline.

Note that the measure is not recording a nominal stress value. Since secondary welds are low-stress, chances are good that the calculated fillet weld size will be impractically small anyway.

The model was meshed and solved after defining the measure using the linear single-pass adaptive solver. In addition to producing stress results that can be rendered in the results viewer, a text summary of the solver execution is also generated. This log contains simulation measure data points in the base units of the model.
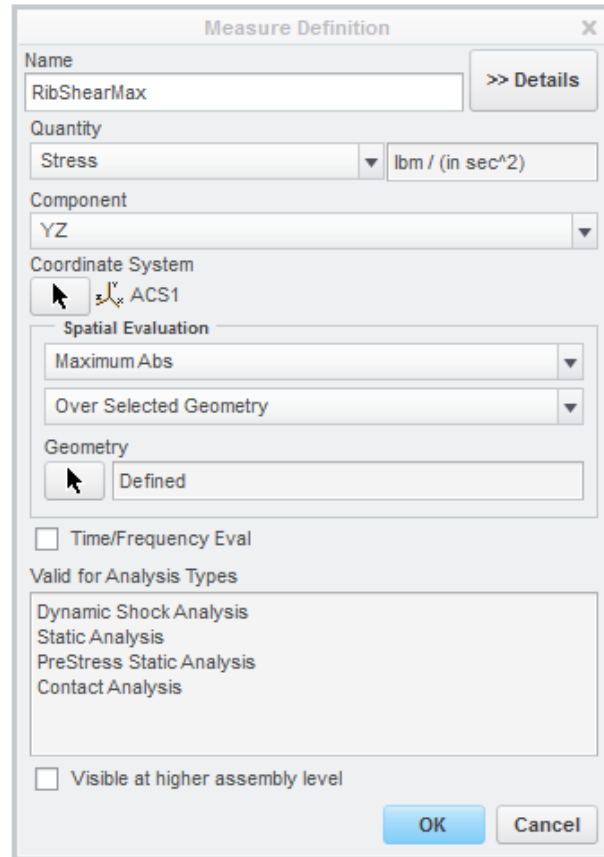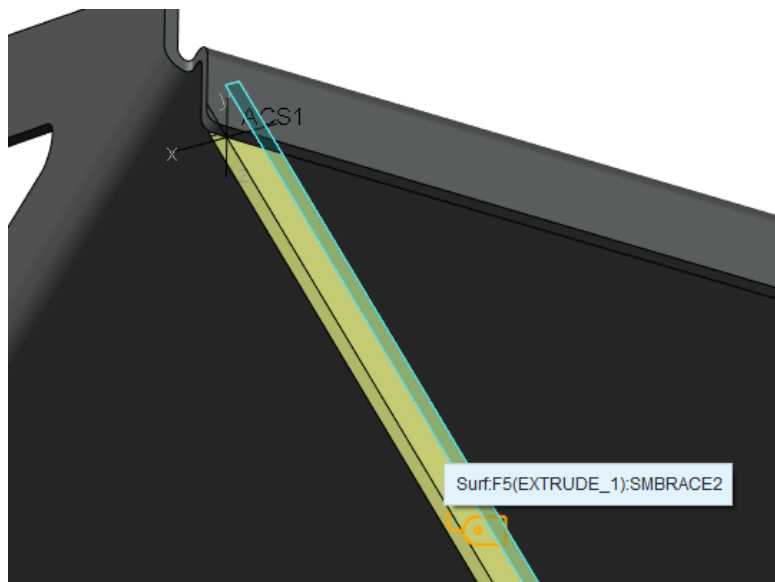
Figure 14: Measure Configuration

Figure 15: Selected Surface

The base units for this model are pounds (mass), inches and seconds. The maximum shear stress at the rib to plate interface was reported as:

$$\tau_{YZ} = 8.268e5 \frac{\text{lb}_\text{m}}{\text{in} - \text{s}^2} \times \frac{1 \, lb_f}{386.4 \frac{\text{lb}_\text{m}}{\text{in} - \text{s}^2}} = 2{,}139 \, psi$$

Since the rib plate is constant thickness, multiplying the shear stress value above by the plate thickness (.125 inch) results in an estimate of shear force per unit length in the rib:

$$\tau_{YZ} = 2{,}139 \, psi \times .125 \, in. = 267.5 \, lb/in.$$

Standard fillet weld sizing equations can be used to calculate the required leg size of the weld. Since this is a low-stress secondary weld, the calculated weld leg size will be very small. Instead, we will use the calculated leg size to calculate parameters for an intermittent (skip) weld to reduce heat inputted to the aluminum. Since the base material is AL5052-H32, the welding process will use 5356 aluminum filler which has a minimum required tensile strength of 38 ksi, and a fillet weld shear allowable of 11.4 ksi will be used (30%).

$$L = \frac{267.5 \, {}^{lb}/_{in.}}{2 \cdot .707 \, in. \cdot 11{,}400 \, psi} = .0166 \, in.$$

At less than two-hundredths of an inch the calculated leg size is impractically small. It'd be unnecessarily difficult and time consuming for a welding fabricator to make a weld this size. A better solution is to employ skip welding with a larger leg size (material thickness in this case). The calculated leg size readily assists in determining an appropriate skip weld ratio:

$$\% = \frac{.0166 \, in.}{.125 \, in.} = 13.28\%$$

The ratio between the weld length and pitch (center-to-center distance between welds) should be 14% or greater. AWS D1.2 requires intermittent fillet welds be no shorter than 1.5 inches, which means the pitch must be 11 inches or less for welds of that length. This strategy does not fit the rib geometry well but would carry the required load. Intermittent fillets of 2 inch in length and on a 7 inch pitch would fit the geometry much better. It places 2 inch fillets at the end of each 16.6 in. long rib, along with a 2 inch weld at the center. The ratio of this strategy is about 28.5%, or twice what's required, but still keeps welding heat input, time and difficulty within practical bounds.

Summary
Measures can allow for quick extraction of necessary stress values from a structural model and are useful for sizing secondary fillet welds. Using a measure that searches over a surface for a maximum stress value yields a very conservative estimate of weld requirements, potentially leading to excessive welding and reducing the value of the quick analysis technique. In such cases where a designer requires a more reasonable estimate of the nominal forces that a weld needs to transmit, measures can be taken at many points along the path where the weld will be placed. Provisions exist in the software to quickly define measures along a pattern of points, the values of which will be placed into the simulation log when solving the model. Plots and

averages of the measures can be used to obtain further insight into the load a weld needs to transmit.

Conclusion

Several viable strategies exist for using Creo Simulate to perform structural FEA on complex structures. The analysis becomes more complex when a welding code is involved that sets nominal stress limits which the design must respect. Finite element codes produce point-stress values which are by definition not nominal values, so extra work is required to estimate a nominal stress.

Shell and ideal beam simplifications don't provide any method for explicitly modeling welds as part of the geometry but may reduce computation time significantly over a model using 3D solids. Thin-shell and slender beam elements also generally require that the largest outside dimension of the part or span of the beam is no less than ten times the material thickness or largest cross section dimension. These requirements may not always be met by a design's geometry, leading to additional inaccuracies in the models results.

General guidelines to follow when developing FEA models for welded structures in Creo Simulate are:

1. For extremely fast, but perhaps less reliable results, insert measures at points to extract stress values. Measures can be rapidly placed at many points if the points are patterned in the part. The output from a pattern of points can be graphed in a spreadsheet tool or averaged and used to calculate a first-order estimate of welding requirements.
2. Remove all irrelevant geometry in an assembly by suppressing it or deactivating it in a simplified representation. Idealizing part interfaces by carefully fixing degrees of freedom on specific geometric features in an analysis with an isolated assembly model is acceptable, unless the object of the analysis is the interface.
3. For small structures with a few welds, modeling the weld filler as a solid and inserting it into an assembly model requires a manageable amount of extra time.
4. When modeling assemblies with fillet welds or other welds that do not completely penetrate the base material, contacts should be placed at free part interfaces to constraint deformation. Models with contacts must be solved with the non-linear solver which increases solver execution time.
5. Rigid links should be avoided at all costs as a substitute for one of the methods described here for modeling welds. Rigid links don't deform and can impart unrealistic extra stiffness to a structure.

Like bolted joints, welded connections are often over-designed, either as a requirement of a design code or to increase the fabrication practicality of a joint. The methods presented allow an engineer to interrogate a design to ensure the welds can safely withstand applied loads.