ANNALS of Faculty Engineering Hunedoara
– International Journal of Engineering
Tome XIV [2016] – Fascicule 3 [August]
ISSN: 1584-2665 [print; online]
ISSN: 1584-2673 [CD-Rom; online]
a free-access multidisciplinary publication
of the Faculty of Engineering Hunedoara

[1.] Imre Zsolt MIKLOS, [2.] Carmen Inge ALIC,
[3.] Cristina Carmen MIKLOS

COMPUTER AIDED DESIGN OF WELDED ASSEMBLIES CURRENT TRENDS

[1-3.] Department of Engineering and Management, Faculty of Engineering Hunedoara,
University Politehnica Timişoara, ROMANIA

ABSTRACT: Welded assemblies are part of permanent joints most often used in practice. The exceptions are where the material parts that will be joined have properties that do not allow their welding. An essential element of these assemblies design is the strength computation of the weld. From design are known mathematical relationships for dimensioning or verification calculations welds, but for complex structures, their use is difficult. In this sense, this paper shows with Autodesk Inventor Professional how is computer-aided design of welded joints in the corner, 3D modeling or verification calculation. Also for a good knowledge of stresses and deformations distributions of the weld seam in the corner, was made an analysis by FEM with Mechanical Autodesk Simulation program.
Keywords: Welded assembly, Fillet weld design, Stress distribution, Finite element

1. INTRODUCTION

Welding is a process for the permanent joining of two or more metal bodies, achieved by cohesion interatomic forces that arise under certain conditions, marginal between atoms belonging to those bodies. Welding is used as a means of assembly, manufacturing process, combined with other processes (forging, splintering, etc.), or as a means of enforcing repair and reconditioning of the worn parts. Welds resistance calculation requires knowledge of the following sizes:

≡ **Tensions allowable** of welded seams, they are determined by the base components and the filler material, the technological process of heat treatment and subsequent mechanical, being generally lower than the resistance of the base material. Thus, allowable stresses of welds is determined by the relation 1, [11]:

$$\sigma_{sa} = \varphi \cdot \sigma_a \tag{1}$$

where: σ_a - is the minimum allowable stress of welded pieces; φ - Reduction coefficient that depends on the welding method and the nature of the stress, $\varphi \leq 1$;

≡ **Sectional dimensions of calculation** $A_s = a \cdot l$, for corner welding shall be determined so:

~ *a* represent the weld height, is calculated as the height of the isosceles triangle that can be placed in the section cordon, for convex and righteous welds a = 0,7s, and for concave weld a = 0,5s (Figure 1), *s* being the minimum thickness of welded parts, [11]

~ l is the length of calculation, defined by the relation 2, (ls - the actual length of the weld), [11]:

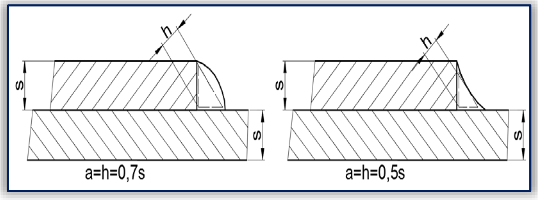

Figure 1. Sectional dimensions for calculation

$$l = l_s - 2a \tag{2}$$

The resistance calculation for fillet welds consists in determining an equivalent tension, according to the relation 3, taking into account the stresses that may occur.

Depending on the nature of the stress in welded seams sections, can be appear normal stresses (stretching or compressive), which acts perpendicular to the direction seam ($\sigma_{\perp}$) and tangential direction perpendicular to the seam ($\tau_{\perp}$) and parallel with it ($\tau_{//}$) as shown in Figure 2, [11].

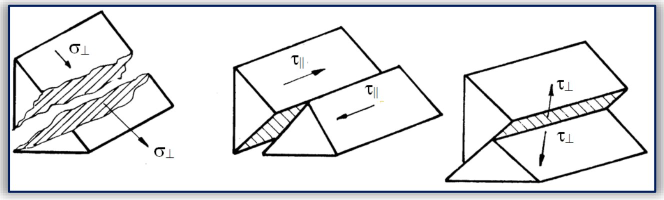

Figure 2. Tensions in the seam welding

$$\sigma_{s,ech} = \sqrt{\sigma_{\perp}^2 + \lambda(\tau_{//}^2 + \tau_{\perp}^2)} \qquad (3)$$

2. 3D MODELING AND WELDING ASSEMBLY CALCULATION

For the problem study is proposed a welded structure consisting of three elements, made by 4-seam weld, the arrangement of their size, loading and the constraints being according to Figure 3. The seam structure loading is carried out by force F = 5000 N.

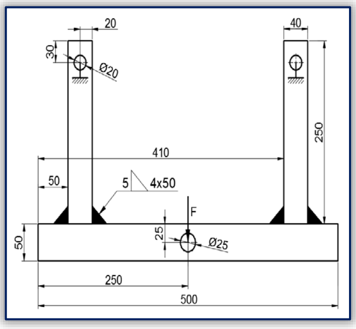

Figure 3. Welded assembly

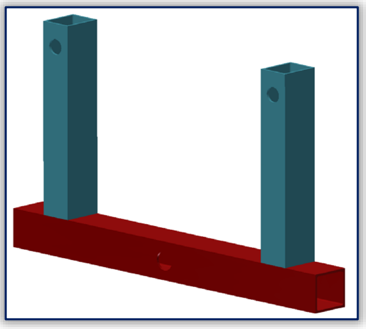
Figure 4. Structure 3D model

In order to achieve the resistance calculation of welded structure, computer assisted, with Autodesk Inventor Professional, it requires 3D modeling of the assembly itself. In this sense in first phase are modeled components elements of the assembly. After applying assembly constraints will result the final structure, shown in Figure 4.

Welded seams are modeled with Fillet weld tool, shown in Figure 5. The options found in the dialog box include: choosing the weld contour which can be plane, convex or concave; choosing the weld size, can be specified by the sides of the right triangle, or its height. Also with the option Start/End Offset can be modeled a weld that does not start exactly at the edges of welded pieces, and with the option Intermittency can model a weld intermittently by specifying two of three parameters which can define: weld length, free space length or number of welds, [4]. Welded structure in this way is modeled as shown in Figure 6.

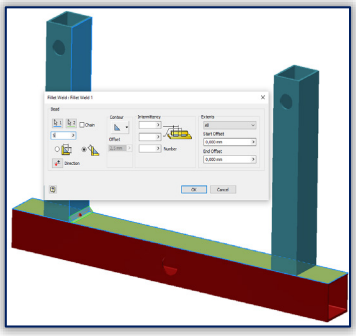
Figure 5. Weld seam modelling

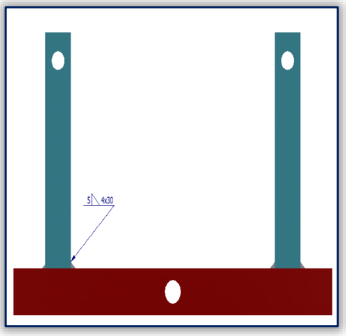

Figure 6. 3D welded structure

Actual strength calculation was performed using the instrument Fillet weld computer, by choosing the calculation as close to the situation studied. This calculation involves the insertion dimensions, loads, respectively weld material choice, according to the interface from Figure 7. The calculation was made for a single weld, therefore the load was distributed properly, [4]. The results obtained from the calculation of strength are shown in Table 1, extracted from the report generated by the application.

Table 1. The results obtained from the calculation of strength

Allowable Stress	σ_A	160,714 MPa
Min. Weld Height	a_{min}	3,609 mm
Resulting Stress	σ_R	116,000 MPa
Safety Factor	n_s	1,400 ul
Max. Acting Force	F_{max}	1731,835 N
Check Calculation		Positive

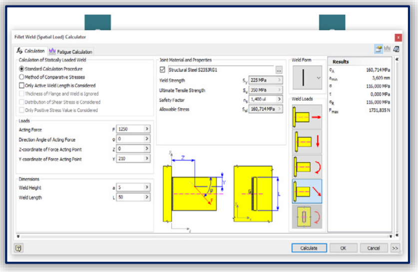
Figure 7. Corner weld calculation

According to the results shown in Table 1, the weld stress is less than the admissible ($\sigma_R < \sigma_A$), or the maximum height of the weld is lower than that used (a_{min} < 5 mm), so the strength calculation is satisfactory. Given that the sizes used in calculating the stress are not exact, and calculation variant chosen is not identical to the charging scheme, for higher accuracy of results required an analysis by finite element method for welded structure.

3. WELDING ASSEMBLY FINITE ELEMENT ANALYSIS

FEM appeared as a necessity to study stress state and deformation for stress structures of high geometric complexity where the calculation is made easier if the whole is divided into easier areas. FEM is a mathematical method of numerical integration for equations with partial derivatives as a variational form. I mean, all the problems of calculation for elastic structures will be reduced to a system of partial derivatives equations, which, often cannot be solved analytically. This deficiency has been eliminated by using digital computers, which ensures rapid means of making a large volume of calculations involved in finite element analysis and made practical the method to be applicable. Together with the development high-speed digital computers, application of finite element method has progressed with impressive speed high [15].

To determine the stress state was chosen Static Stress with linear material model variant, that is used when you want to calculate displacements and stresses due to static loads, size or direction of loading will not change over time, mass model can be used to determine loads such as gravity, respectively when the edge conditions does not modify [18].

Analysis was performed with Autodesk Simulation Mechanical software using the following steps:

~ Importing 3D geometric model obtained under paragraph 2

~ choosing the analysis type - Static Stress with linear material model

Static Stress with Linear Material Models may have multiple load cases. This allows a model to be analyzed with multiple loads while solving the equations a single time. The following is a list of load case multipliers that were analyzed with this model (table 2).

Table 2. The list of load case multipliers

Load Case	Pressure/Surface Forces	Gravity/ Acceleration	Angular Velocity	Angular Acceleration	Displacement	Thermal	Electrical
1	1	1	0	0	1	0	0

~ geometric model meshing (figure 8)

Meshing is fundamental demarche required by MEF and is to switch from continuous structure (with an infinite number of points) to a discrete model, with a finite number of points (nodes). Brick type element is used, which are elements of four, five, six or eight nodes made in three-dimensional space. Brick-type items, by definition, can't have rotational degrees of freedom so they will be used when the model have been applied forces, not moments [18].

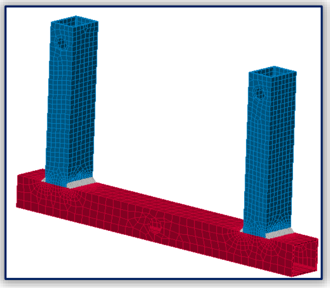

Figure 8. Model meshing

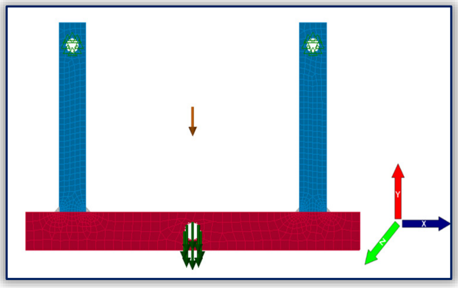

Figure 9. Defining boundary conditions and loadings

~ choosing materials for component parts, including welds

~ Defining edge conditions, it opted for surfaces restraint for the 4 holes from the vertical pieces with General Surface Constraint option, as specified in Table 3 - those surfaces, respectively constrained degrees of freedom.

Table 3. General Surface Constraint options

ID	Part Number	Surface Number	Tx	Ty	Tz	Rx	Ry	Rz
1	7	9,10	Yes	Yes	Yes	Yes	Yes	Yes
3	6	9,10	Yes	Yes	Yes	Yes	Yes	Yes

Table 4. Surface Force options

ID	Part Number	Surface Number	Magnitude (N)	Vx	Vy	Vz
1	5	9,10	-5000	0	1	0

~ Defining uploads, was chosen the force distributed on the holes surface from the horizontal piece, with Surface Force option, as specified in Table 4 these areas, respectively direction of the applied force.

In defining loads was taken into account the own weight of the structure. The acceleration/gravity direction multiplier is multiplied by the acceleration Due To Body Force, which is then multiplied by the acceleration/gravity load case multiplier (table 5). Acceleration Due To Body Force = 9814.56 mm/s^2.

Table 5. The acceleration/gravity direction multiplier

Acceleration/Gravity X Multiplier	Acceleration/Gravity Y Multiplier	Acceleration/Gravity Z Multiplier
0	-1	0

Figure 9 shows the structure that will be analyzed, with margin conditions and loads defined.

- Defining the contact type between the parts, it opted for a contact surface between the structure elements and welded contact between the welds and structure elements
- Realize the analysis and interpretation the results

Following the results of the analysis can be visualized such as von Mises stress status (Figure 10), or the deformations status (Figure 11). It is noted that maximum stress has the value 152,43 N/mm², less than that allowable, 160,7 N/mm² (Table 1) for the used material. In Figure 11 we can see a detail of a weld zone, together with the contact zone between elements.

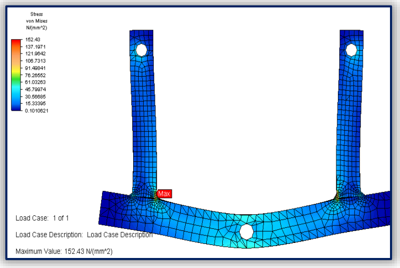

Figure 9. Stress status

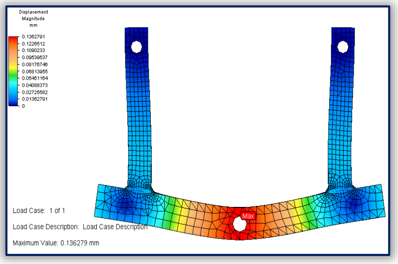

Figure 10. Deformations status

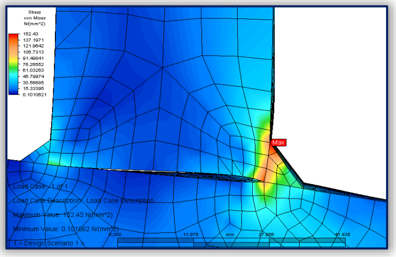

Figure 11. Maximum stress

4. CONCLUSION

The results obtained with Autodesk Inventor Professional are satisfying, but for better accuracy thereof, it has opted for a finite element analysis, accomplished with Autodesk Simulation Mechanical. In this case, maximum stress obtained is higher than in the previous case, 116N/mm ² (Table 1) but below the allowable limit for the used materials. In conclusion, computer aided design of welded structures made by welding corner is an easier, interactive and effective method.

References

[1.] Al-Samhan, A and Darwish, S.M.H: Finite element modeling of weld-bonded joint, Journal of Materials Processing Technology Volume 142, Issue 3, 2003, pp. 587–598

[2.] Barsoum, Z and Lundbäck, A: Simplified FE welding simulation of fillet welds – 3D effects on the formation residual stresses, Engineering Failure Analysis, Volume 16, Issue 7, 2009, pp. 2281–2289

[3.] Bursi, O.S and Jaspart, J.P: Calibration of a finite element model for isolated bolted end-plate steel connections, Journal of Constructional Steel Research Volume 44, Issue 3, 1997, pp. 225–262

[4.] Cioată, V.G; Miklos, I.Zs: Proiectare asistată de calculator cu Autodesk Inventor, Editura Mirton Timişoara, 2009

[5.] Doerk, O; Fricke, W and Weissenborn, C: Comparison of different calculation methods for structural stresses at welded joints, International Journal of Fatigue, Volume 25, Issue 5, 2003, pp. 359–369

[6.] Dolga, L: Parametric and feature – based modelling with application in Catia and Inventor, Editura Politehnica, Timişoara, 2004.

[7.] Faur, N., Elemente finite, Editura Politehnica, 2002

[8.] Gafiţeanu, M: Organe de maşini, vol I,II, Editura Tehnică Bucureşti, 2001

[9.] Girão, A.M; Simões da Silva, C.L and Bijlaard, F.S.K: Finite-Element Modeling of the Nonlinear Behavior of Bolted T-Stub Connections, Journal of Structural Engineering Volume 132, Issue 6 2006, pp. 918-928

[10.] Gurney, T.R: Finite - element analyses of some joints with the welds transverse to the direction of stress, Welding Research International Volume 6, Issue 4, 1976, 20 p.

[11.] Miklos, I: Organe de maşini. Organe de asamblare, Editura Mirton, Timişoara, 2001

[12.] Muhs, D: Roloff/Matek, Organe de maşini, Vol. I, II, Editura MatrixRom, Bucureşti, 2008

[13.] Nishikawa, H; Serizawa, H and & Murakawa, H: Actual application of FEM to analysis of large scale mechanical problems in welding, Science and Technology of Welding and Joining, Volume 12, Issue 2, 2007, pp. 147-152

[14.] Pang, H.L.J: Analysis of weld toe profiles and weld toe cracks, International Journal of Fatigue, Volume 15, Issue 1, 1993, pp. 31–36

[15.] Spyrakos, C.Ch: Finite element modeling in engineering practice, Morgantown, W. Va.: West Virginia University Press, 1994.

[16.] Teng, T.L; Fung, C.P; Chang, P.H and Yang, W.Ch: Analysis of residual stresses and distortions in T-joint fillet welds, International Journal of Pressure Vessels and Piping, Volume 78, Issue 8, 2001, pp. 523–538

[17.] Yang, J.G; Murray, T.M and Plaut, R.H: Three-dimensional finite element analysis of double angle connections under tension and shear, Journal of Constructional Steel Research, Volume 54, Issue 2, 2000, pp. 227–244

[18.] * * * Finite Element Analysis in Practice - Instructor Manual/University Course Curriculum, Algor, Inc. Pittsburgh, USA