12th ANKARA INTERNATIONAL AEROSPACE CONFERENCE **AIAC-2023-036**
13-15 September 2023 - Online, Ankara, TURKIYE

DEVELOPMENT OF A DESIGN TOOL FOR OPTIMIZATION OF THE NUMBER, SIZE, AND FLANGE GEOMETRY OF THE BOLTS USED IN ASSEMBLING CYLINDRICAL BODIES

Zekeriya Buğra Köksal[1]
Roketsan Missiles Inc.
Ankara, Turkey

Erdem Acar[2]
TOBB University of Economics and Technology
Ankara, Turkey

ABSTRACT

In this study, a bolt design optimization tool is developed for optimization of the flange geometry and the bolts used in cylindrical structures. Firstly, a bolt design tool is developed by using classical bolt-member theories and NASA procedures, since the code is aimed to be used in rocket bodies. The accuracy of the developed design tool is validated by comparing it to finite element analysis using ABAQUS. Then, an optimization tool is built by using MATLAB. Finally, the performance of the bolt design optimization tool is validated using example problems in the literature and a user interface is designed for the code.

INTRODUCTION

Cylindrical bodies such as rockets are often used, and these bodies cannot be produced in one piece for the most part. For this reason, it is important to keep more than one body together decently. This process of holding together may require decoupling again in some critical situations. The best mechanical connection that meets these requirements is a bolted connection (see Fig. 1).

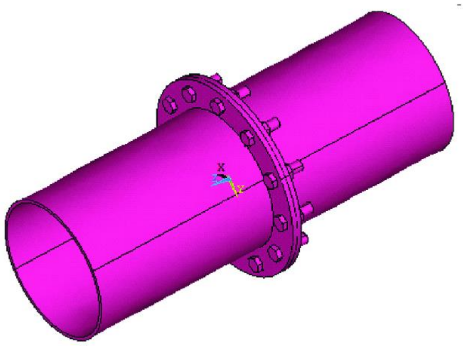

Figure 1. A representative image of one of the areas that the code can be used [Zhi-Jie Wan, 2017]

[1] Design Engineer, Email: bugra.koksal@roketsan.com.tr
[2] Prof., Department of Mechanical Engineering, Email: acar@etu.edu.tr

Bolts have been used as mechanical fasteners for years due to their ability to be disassembled and their established system. The number and properties of bolts required for use are generally determined by hand calculations, but there is a high margin of error if these calculations are not validated by using high-fidelity analysis. However, a high-fidelity analysis is often costly due to the definitions of pre-loading and contact.

In this study, a design optimization tool is developed that can determine the required number of bolts, properties, and flange geometry for the assembly of cylindrical parts. The developed code is validated by high-fidelity finite element analysis. The developed code could be used in the assembly of cylindrical bodies such as rocket bodies to save time and cost.

Various formulas are available for bolt calculations in the literature, in particular NASA reports. These formulas are based on load sharing, which is created by connecting the stiffness of bolts and flanges (see Fig. 2), and then to establish the relationship between their strength [Jeffrey A. Chambers, 1995 and P.J. Crescimanno, 1981].

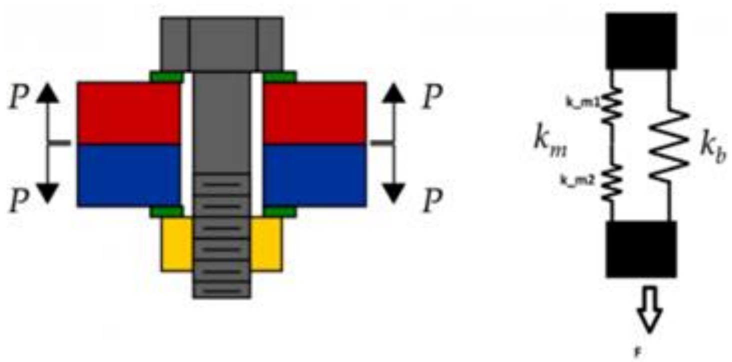

Figure 2. A representative image of basic stiffness effect in bolt calculations [Bickford and John H, 1998]

The relationship between member and bolt can also be observed by bolted joint diagrams, and one example is shown in Figure 3. This graph shows that bolt extension and member compression are the same in terms of distance, however the required forces are not the same due to stiffness differences. The higher stiffness requires higher force to extend or compress the same distance.

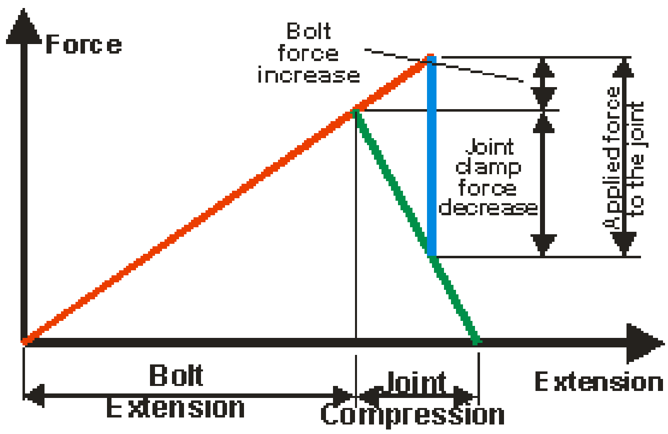

Figure 3. A representative image of the bolted joint diagram [Bickford and John H, 1998]

Bolt and member, after applying preload, create a pressure cone as shown in Figure 4. This figure demonstrates that the interface between two members carries the most pressure while the top and bottom faces of the members carry least. The pressures at the top and the bottom faces do not have to be the same, depending on the thicknesses of members.

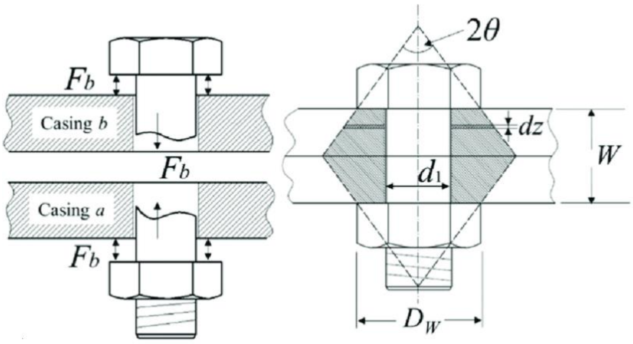

Figure 4. A representative image of the pressure cone [Hehe Kan and Zhi-Min Li, 2020]

METHOD

Optimization algorithm

In optimization, two MATLAB built-in functions are used: “gamultiobj” and “fmincon”. The “gamultiobj” function is based on genetic algorithm and it can readily solve multi-objective optimization problems. The “fmincon” function is a gradient-based optimizer, developed originally to solve single-objective optimization problems, but it can also solve multi-objective optimization problems. In this study, there are two objective functions, as will be discussed in the next sub-section, and the Pareto front, which is the set of all most efficient solutions (see Fig. 5), is generated as the solution of the optimization problem.

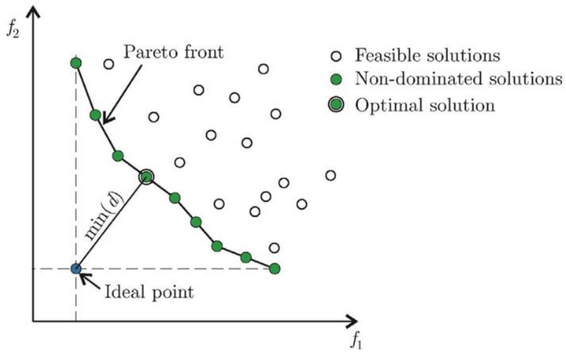

Figure 5. A representative image of Pareto Front [Facundo Bre and Victor D. Fachinotti, 2017]

Optimization Problem in the Standard Form

The optimization problem of interest can be written in standard form as

Find **x** = [N, D, L] (1)

Min {M(**x**), N(**x**)}

S.t. MS (**x**) ≥ 0.1

0.1 ≤ phi(x) ≤ 0.4

where x is the set of design variables, N is number of bolts, D is diameter of bolts, L is the grip length of the members, M is the mass of the whole system, margin of safety is a type of yield criteria and is entered in the code from the user, phi is the bolted joint stiffness coefficient.

Design variables

In this study, the number of bolts, the diameter of bolts and grip length of members are used as the design variables.

Objective Functions

In this study, there are two objective functions. The first objective function is the mass of total bolt-flange system. Obviously, the flanges lead the total mass since the thickness is critically important, which is a surrounded bulkhead. If the number of bolts is small, then the mass of the bolts is not comparable to that of flanges. The second objective function is aimed to reduce the assembling time required to mount the whole bolt-flange system. Therefore, the second objective is chosen as the number of bolts used in the system.

Constraints

The classical bolt-member relationships are defined in the developed code. These include the stiffness ratio, the preload calculations, and the external loading distribution. After this relationship is established in the code; strength, geometric constraints and bolt standards in the literature are applied as optimization inputs. As a strength restriction, the Margin of Safety method used by NASA in bolt design is applied. The margin of safety (MS) constraint is given in Eq. (1)

$$MS = \frac{1}{\sqrt{(R_t + R_b)^2 + (R_s)^3}} - 1 \tag{2}$$

where R_t, R_b and R_s are the coefficients obtained from the axial, bending and shear loads on the bolt, respectively. Using Eq. (2), a combination of the highest loads that can be applied on the bolts can be determined. In addition, on the contrary, if the loads are known, the closeness of the bolt stresses to its strength value can be calculated. Usually, MS is desired to be higher than 0.1 [2]. After stress-strength ratios are built, geometrical constraints are taken into account. For example, bolts cannot be so close to each other when perfectly aligned, or to the edges of the flanges.

$$R_t = \frac{P_{0,max} + \text{SF.n.}\phi.F_{ex}}{\text{Tension Allowable}} \tag{3}$$

$$R_b = \frac{\text{SF.M}}{\text{Bending Allowable}} \tag{4}$$

$$R_s = \frac{\text{SF.V}}{\text{Shear Allowable}} \tag{5}$$

The parameters R_t, R_b and R_s can be calculated with the formulas above. They are the ratios of the external forces to the strengths of the bolts. They all must be in the range from 0 to 1 in a well-designed system [1].

The Inputs of the Code

The material properties are required in the code, which include the elastic modulus, strength, and the density. Another input is the outer diameter, which is the only geometrical input of the code. Safety factor and margin of safety are also required to enable the user to change the safety level as desired. The main inputs for the code are, of course, the external loads on the system. They could be tensile force which tries to disassemble the system and creates tension on the bolts. The compression force reduces the preload on the bolt and squeezes the flanges. The shear force creates transverse load and shear stress on the bolts, which is not desired in a usual bolted joint. Last load is the bending moment on the system, which creates both tension and compression on the flange, based on where the moment applied to the system.

Outputs of the Optimization Code

The optimization code works with two different algorithms. Figure 6 presents a comparison of the Pareto Front obtained from these two different algorithms using the inputs given in Table 1. It is seen that the two different optimization algorithms provide similar results. All outputs except Pareto Front, which can be seen in Figure 6, will be explained in the following chapters.

Table 1. Random inputs for an example of the output of the code

Input value	**Unit**	**Meaning**
F=300000	N	Axial tension force
V=40000	N	Shear force
M=20000	N.m	Bending moment
Dr=1.3	m	Outer diameter
SF=2	-	Safety factor
MS=0.1	-	Margin of safety
Member_density=2800	kg/m^3	Flange density
TS=940	MPa	Yield strength of the bolt
Ej=70	GPa	Member elastic modulus
Eb=210	GPa	Bolt elastic modulus

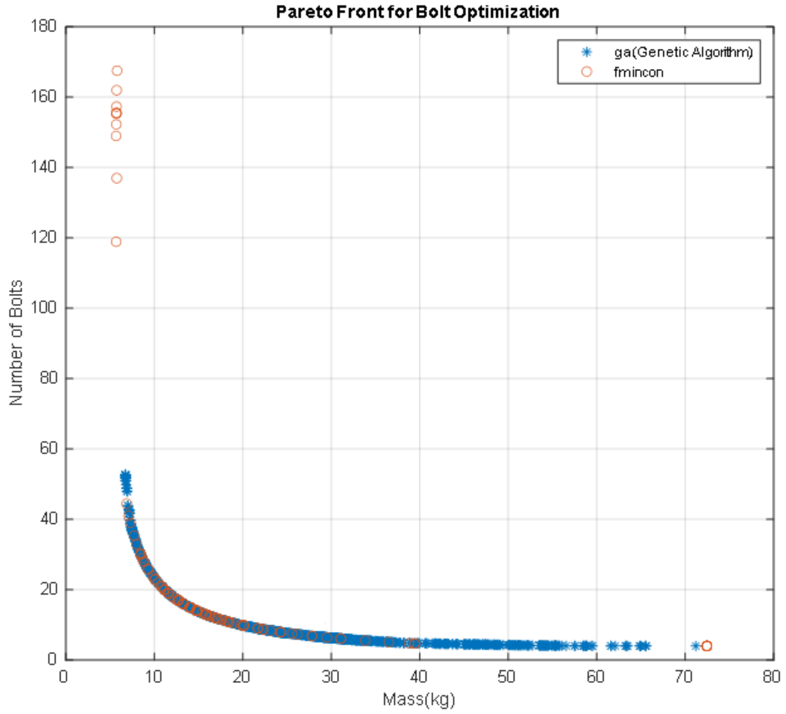

Figure 6. Pareto Front output example of the code

VALIDATION OF THE BOLT DESIGN CODE USING FINITE ELEMENT ANALYSIS

The bolt design code (i.e., classical bolt calculations) is validated by using ABAQUS finite element analysis (FEA) software. The margin of safety criteria, mentioned in the previous section, is validated in terms of all of its sub-components. The tension force, compression force, shear force and bending moment effects are investigated separately. For example, tension force analysis, which is indicated by R_t in margin of safety criteria, is performed in a simplified model as shown in Figure 7. The left side of the figure shows only the bolt, which is shown as a system in the right side of the figure. As shown in the figure, the system is fixed from the bottom and the force is applied from the top with the reference point.

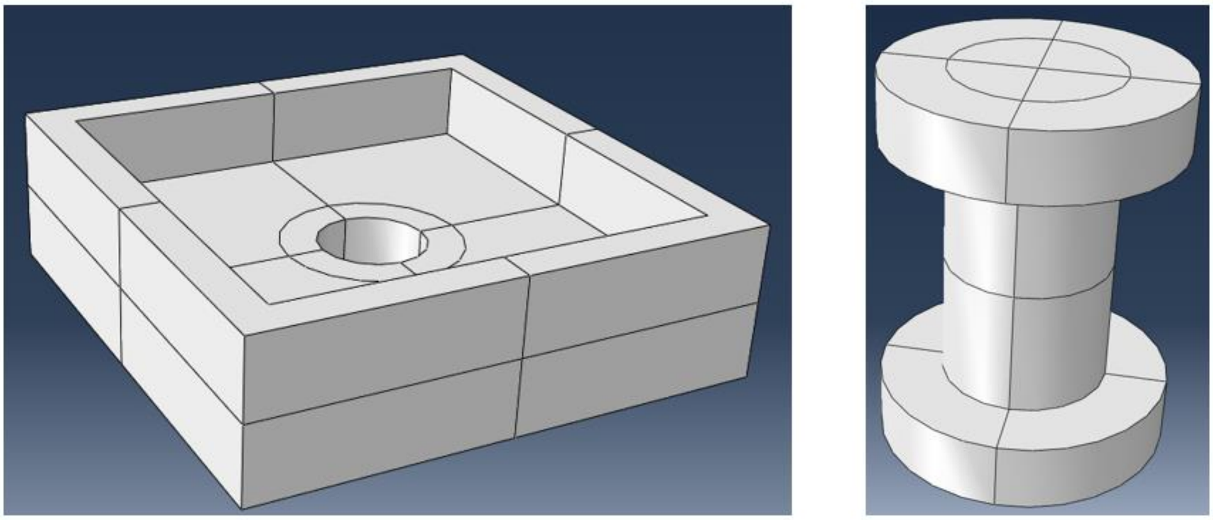

Figure 7. Parts used in FEA(left side symmetrical member, right side bolt-nut assembly)

The validation method starts with proving mathematical parts of the code which provides only stress-strength ratios using the stiffness ratios of the system. A simple case is considered to validate this part, which is described below.

The inputs of the validation case are as follows: F=100 kN tensile force, SF=1.5 safety factor, MS=0.1 margin of safety, Ej=70 GPa member modulus of elasticity, Eb=210 GPa bolt modulus of elasticity. In the analysis, M20 bolt with 15mm flange thickness is used. Using these input values, the output of the code for R_t is 0.78, which indicates that the bolt must be stressed around 78% of its strength. Noting that the yield strength is 940 MPa for a 10.9 class bolt property, the stress in the midline (see A-A in Figure 8) is predicted to be 0.78×940=733 MPa. The FEA results are shown in Figure 8. It is seen that the average stress in the cross-section A-A is 712 MPa. The difference between the analytical prediction of the code and FEA prediction is around 3 percent, and that is an acceptable value in regular bolt analysis.

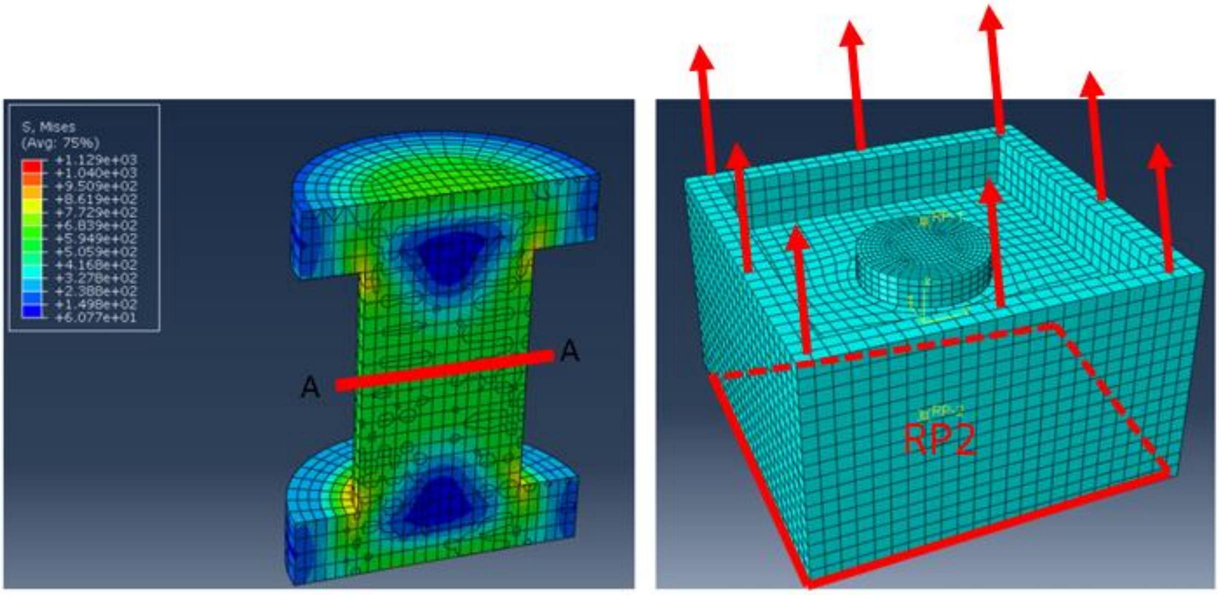

Figure 8. A bolt analysis example of ABAQUS

After performing a comparison in terms of stress prediction, a comparison in terms of mass is performed. The total mass of the system is predicted to be 0.56kg using the analytical code, and 0.57 kg using the FEA. Thus, the difference is around 2 percent, which is considered to be acceptable.

Further Validations

After the tensile load case is proved, the procedure is repeated to increase the accuracy. This time, instead of giving axial load to the system, shear load is given from the RP. Illustration is given in Figure 9.

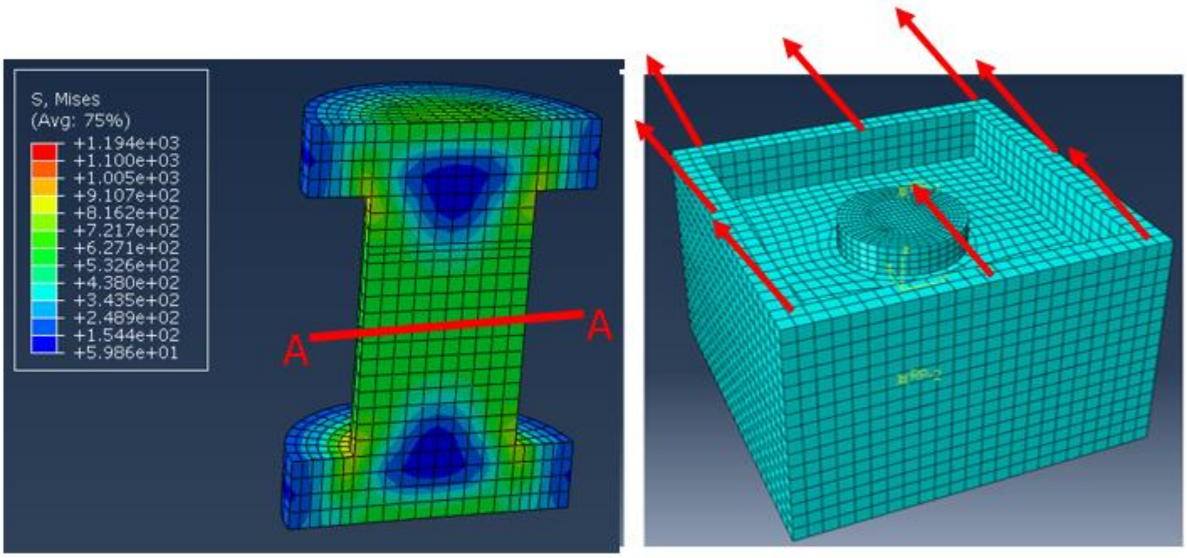

Figure 9. A bolt analysis example of ABAQUS

The inputs of the first validation case are as follows: F=70 kN shear force, SF=2 safety factor, MS=0.15 margin of safety, Ej=210 GPa member modulus of elasticity, Eb=210 GPa bolt modulus of elasticity. In the analysis, the same geometry is used. Using these input values, the output of the code for R_S is 0.69, the stress in the midline (see A-A in Figure 9) is predicted to be 0.69×940=648 MPa. The FEA results are shown in Figure 9. It is seen that the average stress in the cross-section A-A is 635 MPa. The difference between the analytical prediction of the code and FEA prediction is around 2 percent, and that is an acceptable value in regular bolt analysis.

After these validations, the optimization part is validated by using ABAQUS. For validations, the flange-bolt system is examined as a reduced model. First, a load case and material condition is chosen randomly and the outputs of the code are modeled in CATIA. The model is subjected to randomly chosen load case and the results are compared. The reduced model is explained in Figure 10. In every load case, RP is created at the top and bottom of the model. Bottom RP is fixed to the ground and top RP is where the loads are applied to the system. The angles are equal about the bolt, calculated by the half of the distance between the bolts. The load cases are shown separately in the following parts.

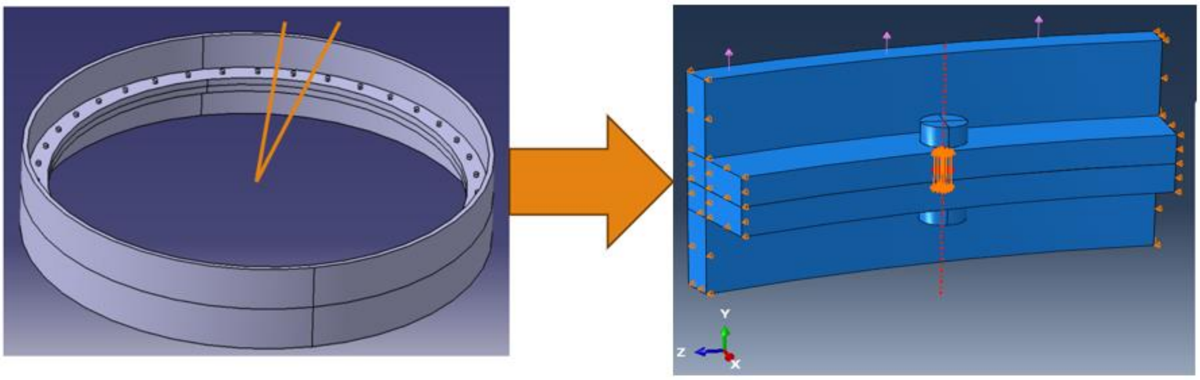

Figure 10. Reduced model explanation

This model is used because of reducing the computational time and cost. The two edges of the model highlighted in Figure 10 is given as boundary condition so that the model acts like it has 360 degrees, as modeled in the left of Figure 10. Every single input set leads to a unique set of output. Every output is modeled separately in ABAQUS as mentioned before. For example, code gives the output set of 32mm flange length, 30 bolts in use and 18mm bolt diameter. All of these parameters are modeled and reduced in one bolt-flange system with every boundary condition added. The procedure is repeated for the randomly chosen conditions below.

<u>Only Tensile Force</u>

The inputs of the code are shown below, the outputs are compared with the analysis results. This comparison will be made in every load case. As can be seen in Figure 11, section A-A(this area will be mentioned as the middle area from now on) has a stress of 580 MPa, and the output R_t of the code is 0.68 for 12 M4 bolts and this means 0.68 x 940 MPa = 639MPa. The difference is 9%.

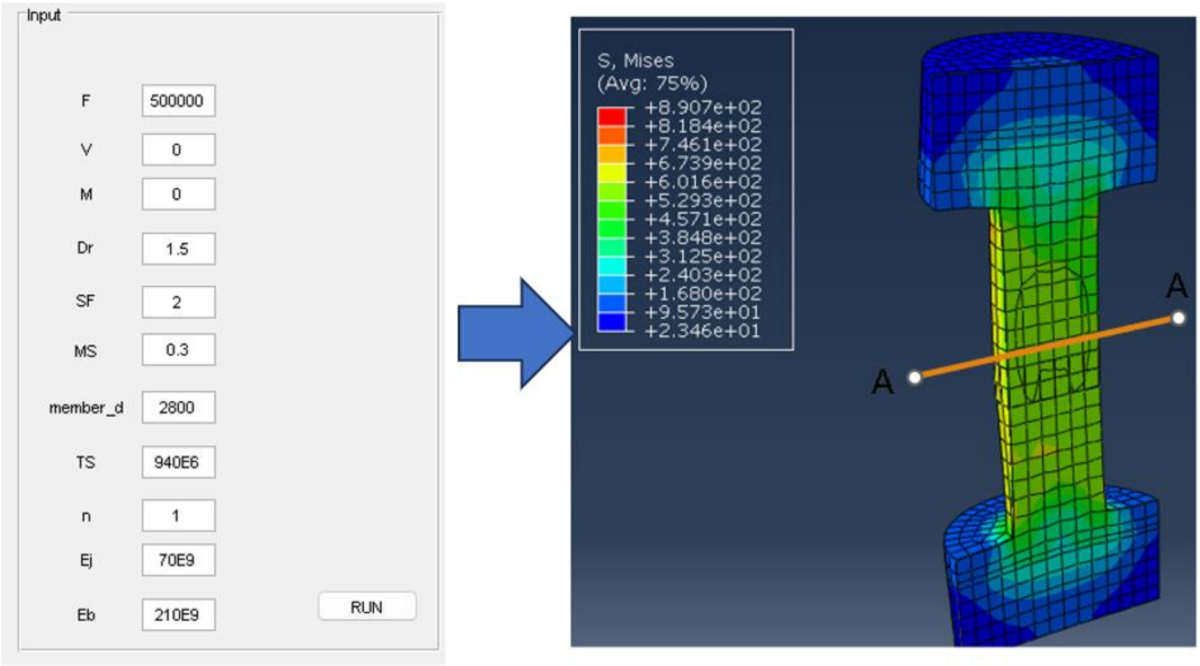

Figure 11. Analysis result of tensile force only

<u>Only Shear Force</u>

The middle area has a stress of 792 MPa, and the output R_s of the code is 0.81 for 24 M6 bolts and this means 0.81 x 940 MPa = 761MPa. The difference is 4%. (see Fig. 12)

Figure 12. Analysis result of shear force only

<u>Combined Loads-1</u>

The combined loadings are the cases both shear and axial forces are applied to the system. The first case has the middle area has a stress of 443 MPa, and the input M_s of the code is 0.3 for 16 M5 bolts and this means 1-0.3=0.7 and 0.7x640=448MPa. The difference is 1%. (see Fig. 13)

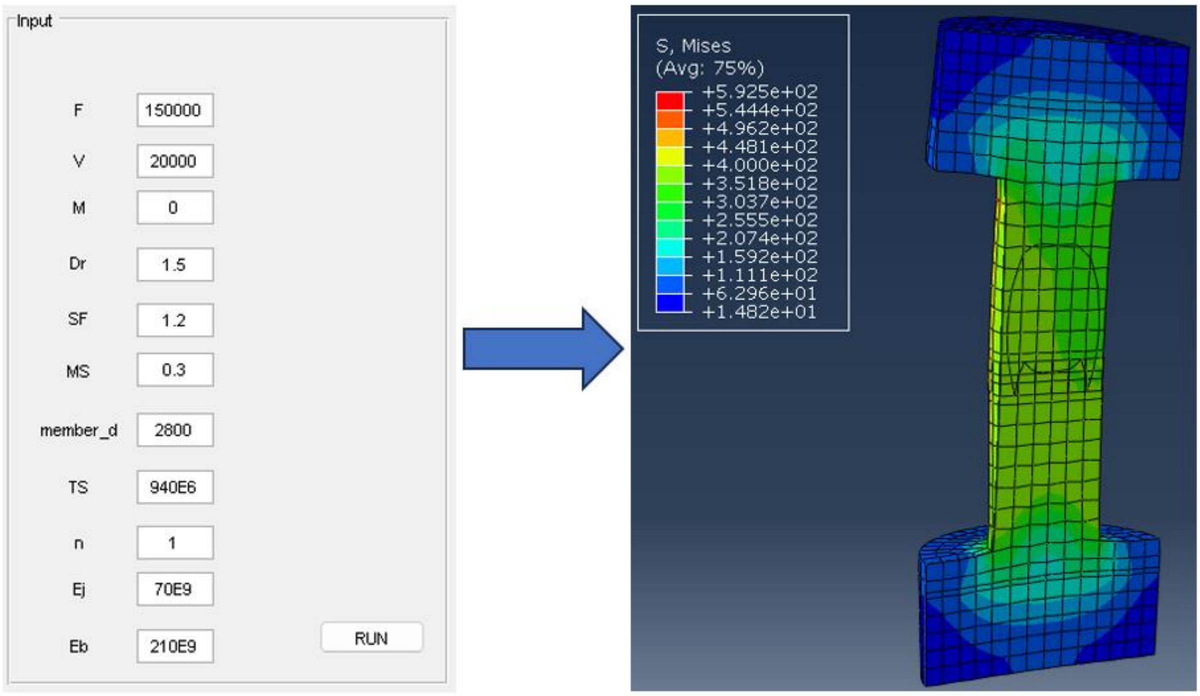

Figure 13. Analysis result of combined loading case-1

<u>Combined Loads-2</u>

This case has the middle area has a stress of 522 MPa. The input M_s of the code is 0.2 for 25 M8 bolts and this means 1-0.2=0.8 and 0.8x640=512MPa. The difference is 2%.(see Fig. 14)

Figure 14. Analysis result of combined loading case-2

<u>Combined Loads-3</u>

This case has the middle area has a stress of 584 MPa. The input M_s of the code is 0.1 for 46 M10 bolts and this means 1-0.1=0.9 and 0.9x640=576MPa. The difference is 2%.(see Fig. 15)

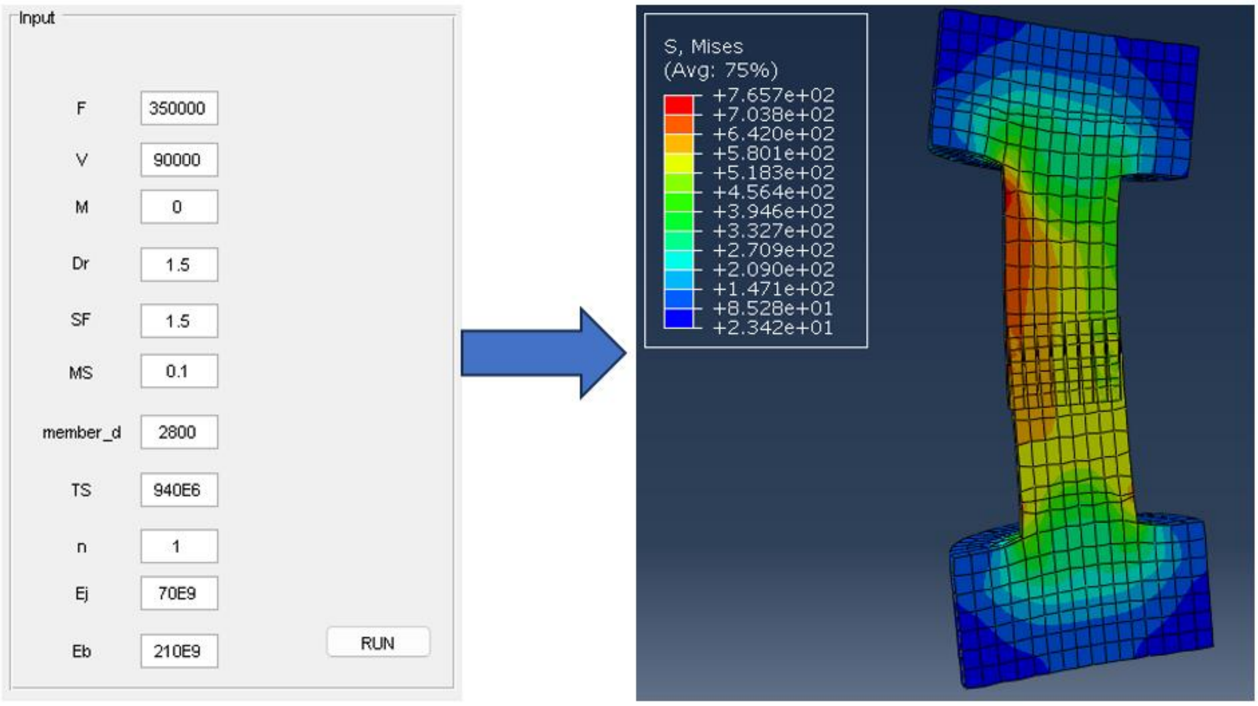

Figure 15. Analysis result of combined loading case-3

<u>Combined Loads-4</u>

This case has the middle area has a stress of 726 MPa. The input M_s of the code is 0.2 for 18 M8 bolts and this means 1-0.2=0.8 and 0.8x940=752MPa. The difference is 4%.(see Fig. 16)

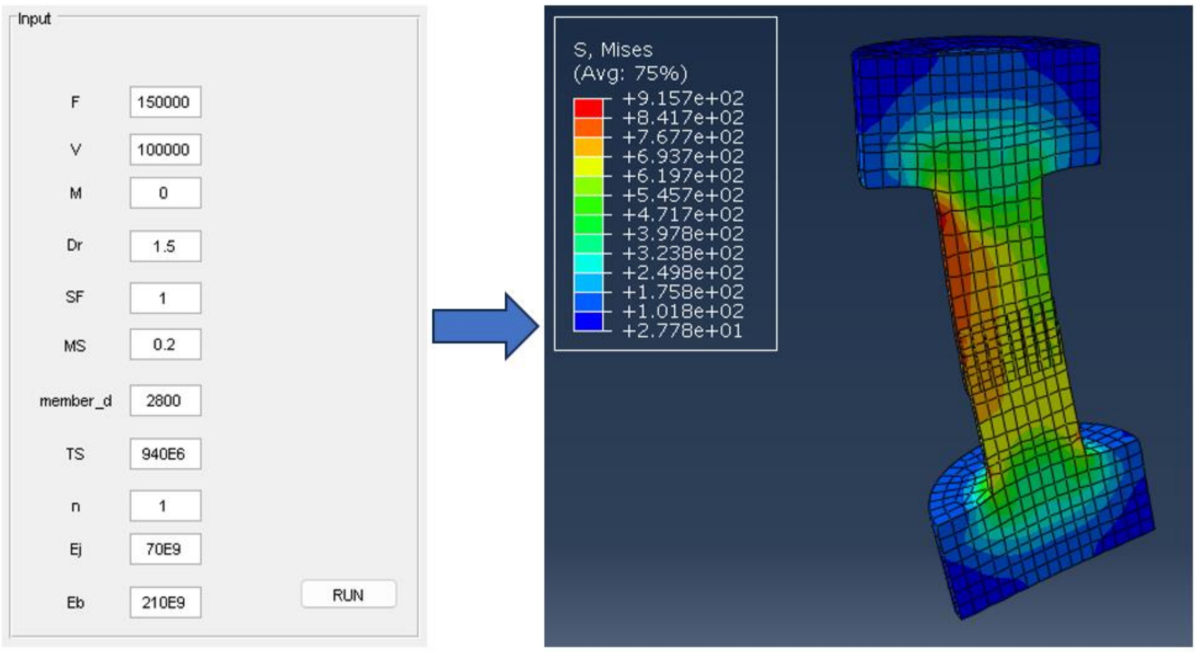

Figure 16. Analysis result of combined loading case-4

<u>Combined Loads-5</u>

This case has the middle area has a stress of 819 MPa. The input M_s of the code is 0.1 for 30 M12 bolts and this means 1-0.1=0.9 and 0.9x940=846MPa. The difference is 3%.(see Fig. 17)

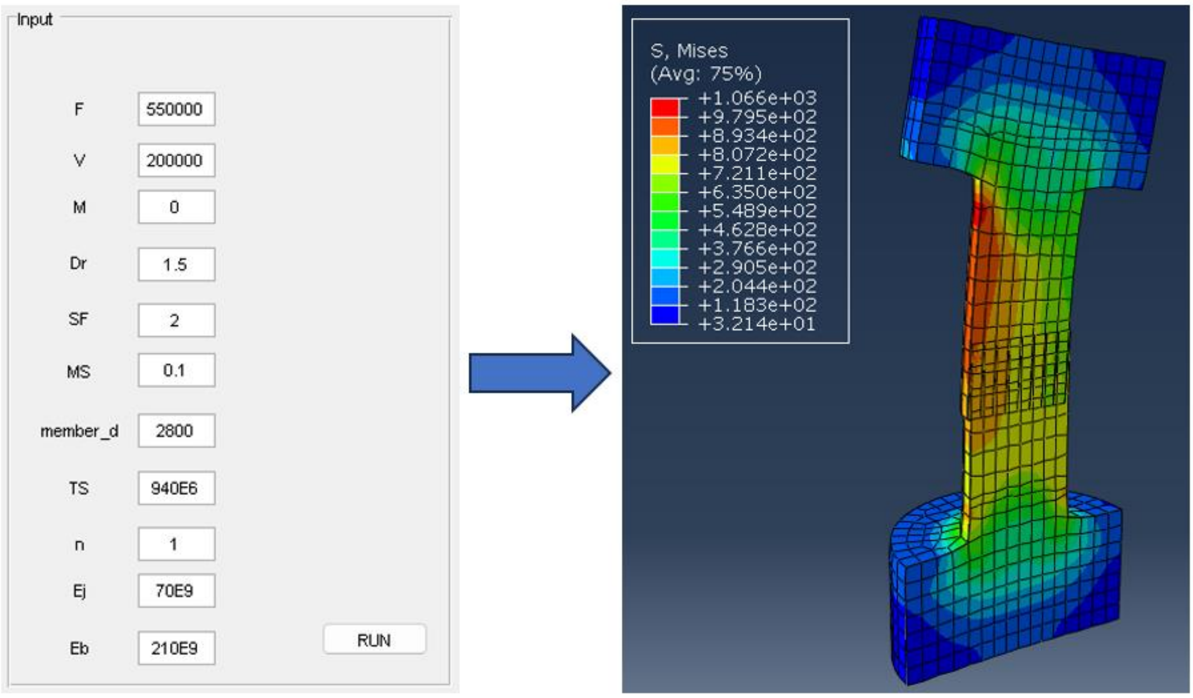

Figure 17. Analysis result of combined loading case-5

<u>Combined Loads-6</u>

This case has the middle area has a stress of 756 MPa. The input M_s of the code is 0.25 for 20 M8 and this means 1-0.25=0.75 and 0.75x940=705MPa. The difference is 7%.(see Fig. 18)

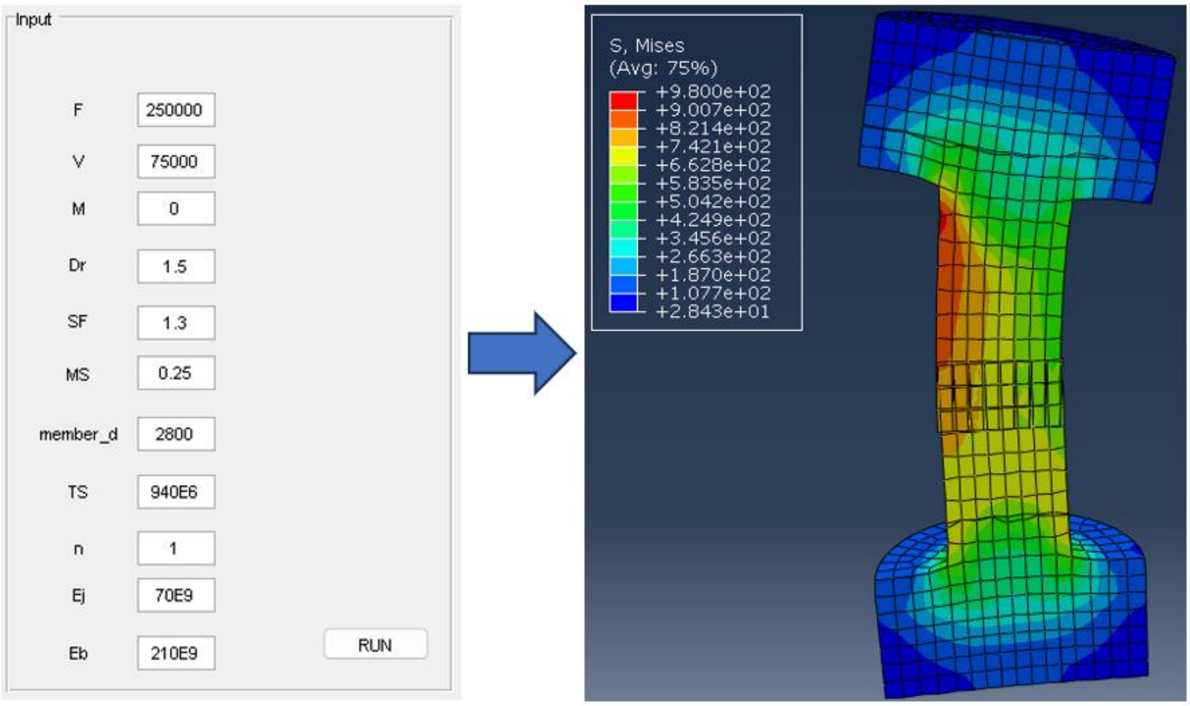

Figure 18. Analysis result of combined loading case-6

USER INTERFACE

A User Interface(UI) is added to the code so that the user of the code no longer needs to enter the inputs in the main code. An example of the UI is explained below in detail. The outputs are descriptive and the Pareto Front is detailed. The inputs are described in the previous parts, outputs are examined below. (see Fig. 19)

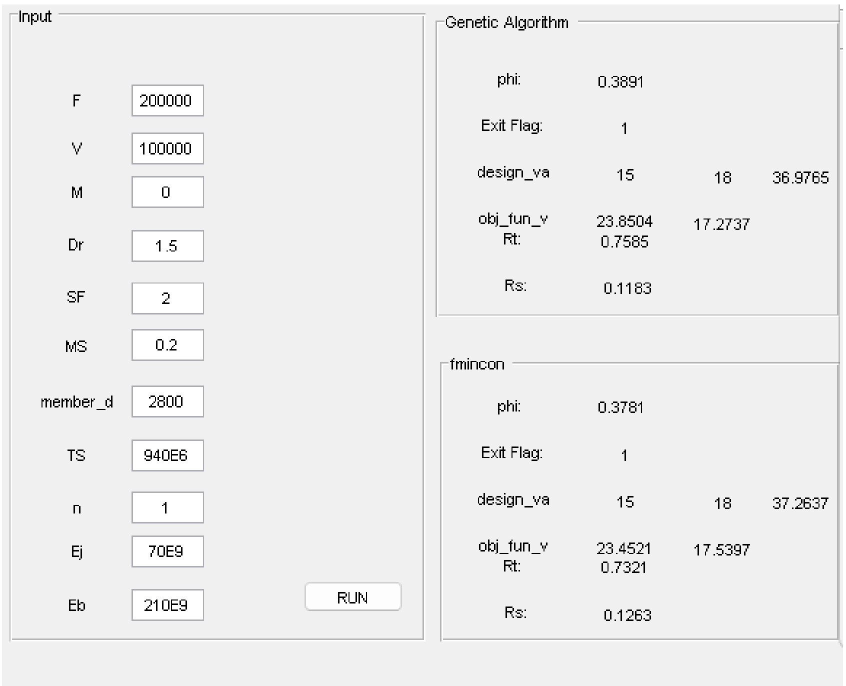

Figure 19. An example of the UI

The outputs are;

- phi: describes briefly of the load distribution between member and the bolt
- exit flag: describes whether the optimization is performed well or not
- design_va: describes the diameter of the bolts, the number of the bolts and the flange grip length, respectively
- obj_fun_v: describes the mass of the total system and the number of the bolts, respectively
- Rt: describes the ratio of the axial forces on the bolt to the yield strength
- Rs: describes the ratio of the shear forces on the bolt to the yield strength
- Pareto Front: describes all possible solutions for the problem

After these validations, a code that can build a optimized flange-bolt system for the given inputs is designed and proved. A user interface is added to the code so that the user of the code reaches every input and output they need in one simplified window. An example of the Pareto Front is explained below.(see Fig. 20)

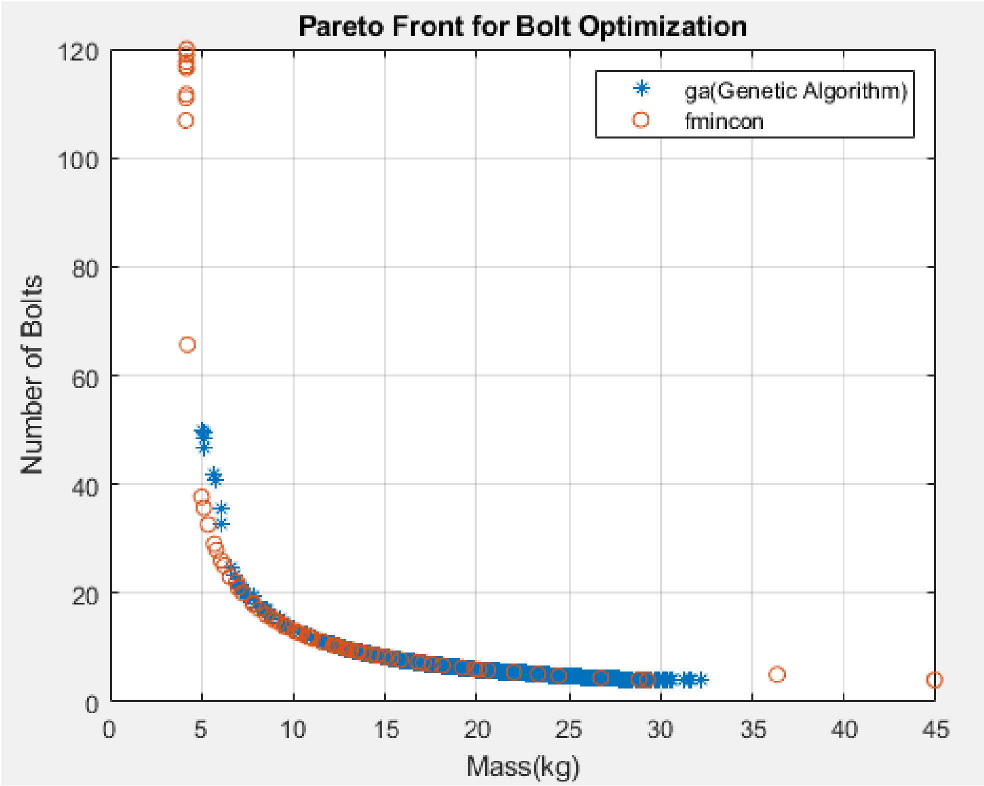

Figure 20. An example of the Pareto Front

This graph shows that, two different optimization technics give similar results. The optimum point where both of the objective functions are optimum is the point that is closest to the origin. After the Pareto Fronts are graphed, analyses are made and the GUI is designed, the aim of the code is achieved.

CONCLUSION

In this study, a flange-bolt design code is built using MATLAB and its optimization toolbox to find optimum parameters such as bolt diameter, bolt number and flange grip length. The code uses the usual bolt-member calculation procedure at first. Then, it optimizes the results for the best possible design parameter combination, using both fmincon and gamultiobj, which are optimization toolbox functions. The validation of the code is divided into two parts, one's aim is to prove usual bolt calculations which is described in NASA joint analysis procedures. This validation provides not only proving the hand calculations but also the analysis methodology constructed is ABAQUS. After the first part of the code is proved, the second part, which describes the optimization, is validated, too. The validations are demonstrated in 10 different load-material cases and 2 different analysis models. The maximum difference between the results and the code outputs is about 9%. After the validations, a user-interface is added to the code. Inputs can be entered and outputs can be seen in one simplified window, including Pareto Front for the two different objective functions.

REFERENCES

Bickford and John H.(1998) *Handbook of Bolts and Bolted Joints*, 9780585158334, June 1998

Facundo Bre and Victor D. Fachinotti(2017) *A Computational Multi-Objective Optimization Method to Improve Energy, Efficiency and Thermal Comfort in Dwellings*, August 2017

Hehe Kan and Zhi-Min Li(2020) *Assembly Research of Aero-Engine Casing Involving Bolted Connection Based on Rigid-Compliant Coupling Assembly Deviation Modeling*, March 2020

Jeffrey A. Chambers(1995) *Preloaded Joint Analysis Methodology for Space Flight Systems*, December 1995

P.J. Crescimanno(1981) "*Forces in Bolted Joints: Analysis Methods and Test Results"* Utilized for Nuclear-Core Applications, March 1981

Zhi-Jie Wan(2017) *Equivalent Simulation of Mechanical Characteristics for Parametric Modeling of Bolted Joint Structures*, June 2017