10th ANKARA INTERNATIONAL AEROSPACE CONFERENCE AIAC-2019-169
18-20 September 2019 - METU, Ankara TURKEY

TOPOLOGY OPTIMIZATION AND STRUCTURAL ANALYSIS AN AIRCRAFT WING

Harun İNCİ[1]
Samsun University
Samsun, Turkey

Zeliha ÇAMUR[2]
Samsun University
Samsun, Turkey

Seher EKEN[3]
Samsun University
Samsun, Turkey

ABSTRACT

In this study, we carry out the topology optimization and structural analysis of an aircraft wing. The first part of this study is devoted to perform the topology and shape optimization of the two-dimensional wing-box structure to obtain the best tradeoff between strength and weight. After this part, the three-dimensional wing design is constructed based on this optimized wing-box shape. For the structural analysis, a finite element code in MATLAB has been developed to model one- and two-dimensional elements and tested with a commercial FE software. After validating our FE code, the static and dynamic analysis of an existing aircraft wing has been performed.

WING-BOX TOPOLOGY OPTIMIZATION

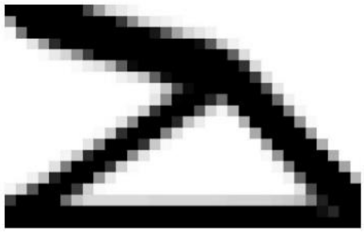
The design objective is defined as eliminating unnecessary materials in the structure with the use of topology optimization to achieve a better load distribution throughout the structure. Topology optimization is a mathematical approach to finding an optimized material distribution in a designated design area. This approach is based on the elimination of other materials by protecting vital load-bearing materials and as a results lightweight and strong structures can be achieved [Walker, Liu and Jennings, 2015]. Due to rising demand for lightweight and strong structures in aerospace engineering, we aimed to use topology optimization to obtain the best design in terms of strength and weight.

Test Case

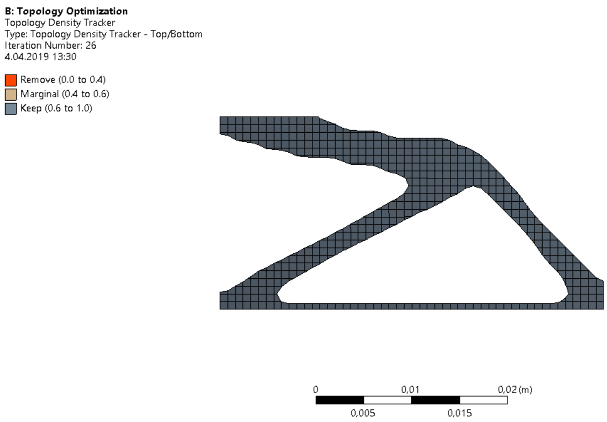
This part of the study explains the details of the topology optimization that is made to enhance strength and reduce weight of the wing structure. Before proceeding the optimization of the wing structure and choosing our topology optimization tools, we first review two tools that can be applied for the topology optimization. The first software is 99 lines optimization code developed in MATLAB [Sigmund, 2001]. The second one is ANSYS Workbench Topology Optimization (TO) tool. For the following cantilevered beam example (Figure 1a), we used both 99lines and ANSYS TO tool and had the results given in Figure 1 (b), (c).

[1] Undergrad. in Aerospace Engineering, Email: haruninci96@gmail.com
[2] Undergrad. in Aerospace Engineering, Email: zeliha.camur4@gmail.com
[3] Asst. Prof. Dr. in Aerospace Engineering, Email: seher.eken@omu.edu.tr

(a)



(b)



(c)

Figure 1: (a) Fixed-free beam example, (b) MATLAB [Sigmund, 2001], (c) ANSYS Topology optimization results

Wing-Box Optimization

When the optimization results are investigated, we observed that the designs optimized using both tools resemble each other. We choose ANSYS Workbench to conduct the topology optimization of the wing. To do this, we have taken an existing wing which is the one of *AT-401 Air Tractor* 's [Janes]. The properties of this aircraft are listed in Table 1.

Table 1: AT-401 Air Tractor's Properties

Wing Span	14.97 m
Mean Aerodynamic Chord	1.83 m
Taper Ratio	1
MTOW	34962.8 N
Wing Profile	NACA 4415

The section of the wing of this aircraft is NACA 4415 Airfoil has a chord of 1.83 m. Using these dimensions, we specified the shaded area in Figure 2 as the design space. The outer surfaces and spar were excluded from the topology area.

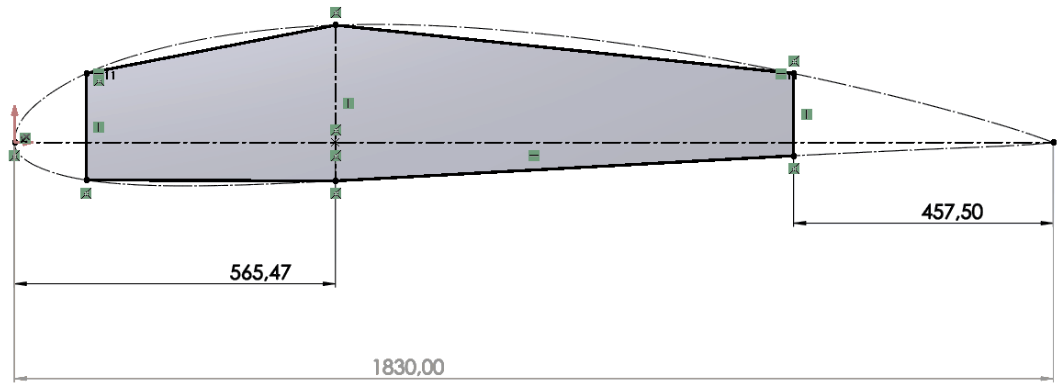


Figure 2: Optimization design space

The topology and shape optimization are made for this design space to maintain a volume fraction of 30 percent. Design limitations are set according to parameters of AT-401 Air Tractor aircraft. The distributed force of 69925.6 N was exerted on the outer surfaces. According to these conditions given in Figure 3(a), the wing box was optimized using ANSYS Workbench TO tool, the optimized space is shown in Figure 3(b). As seen from this figure, this section is determined as our wing-box structure. In order to perform structural analysis, the optimized space is further cleaned and smoothed to finally give the shape in Figure 3(c). This section is set to be the 2-dimensional section of the wing. The design surface obtained according to the optimization results was analyzed by using truss and beam elements in the finite element code developed in MATLAB.

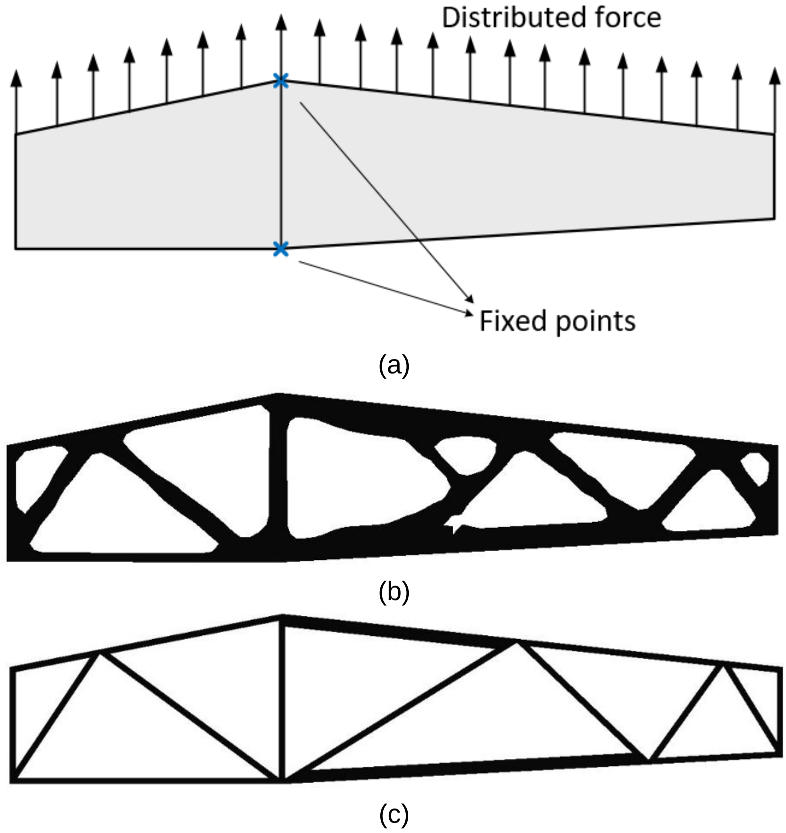


Figure 3: (a) 2-D wing-box (b) Optimized 2-D shape (c) Cleaned, finalized 2-D section.

STRUCTURAL ANALYSIS

The structural analysis is carried out using the numerical code developed in MATLAB. This code is based on finite element modelling and only covers the formulation and simulation of 1-dimensional truss and beam elements. The construction of the FE code is done according to the flowchart shown in Figure 4. As much as the code covers the finite element modelling of the beam elements, our physical interest in the extended abstract is limited to model the wing using only truss elements.

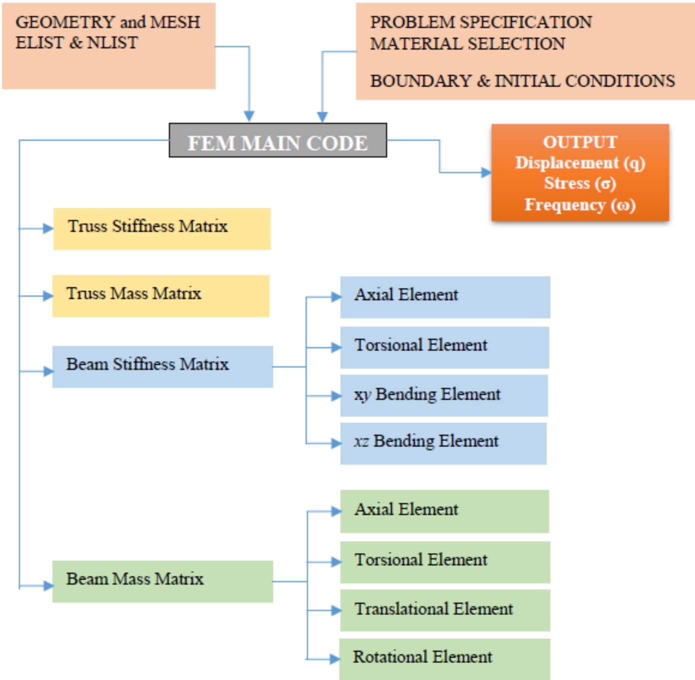


Figure 4: The flowchart of the FEM code

Truss Element

The truss members are widely used in many engineering structures. Examples of usage of these structures are in bridges, aircraft wings, fuselages, cranes. These members can only carry uniaxial loading that is tension and compression. The finite element method has extensively been applied to analyze the structures made up truss members [Logan, 2007].

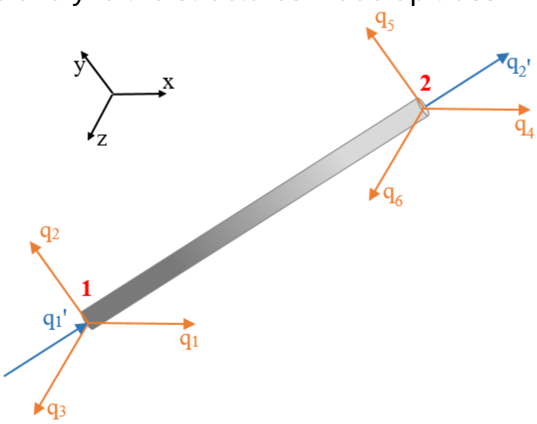


Figure 5:Three dimensional a truss element

The stiffness matrix formulation of the 3-D truss element whose is shown in Figure 5 is given in this section. Node vectors for this 3-D truss element is given as

$$\{q\} = [q_1, q_2, q_3, q_4, q_5, q_6]^T \qquad (1)$$

The transformation between local and global coordinates is made as follows:

$$\{q'\} = [T]\{q\} \qquad (2)$$

The transfer matrix [T] is given by

$$[\mathrm{T}] = \begin{bmatrix} \mathrm{l}_{0\mathrm{x}'} & \mathrm{m}_{0\mathrm{x}'} & \mathrm{n}_{0\mathrm{x}'} & 0 & 0 & 0 \\ 0 & 0 & 0 & \mathrm{l}_{0\mathrm{x}'} & \mathrm{m}_{0\mathrm{x}'} & \mathrm{n}_{0\mathrm{x}'} \end{bmatrix} \qquad (3)$$

$$l_{0x'} = l \quad m_{0x'} = m \quad n_{0x'} = n$$

$$l = \frac{x_2 - x_1}{L_e} \quad ; m = \frac{y_2 - y_1}{L_e} \; ; \quad n = \frac{z_2 - z_1}{L_e} \qquad (4)$$

Thus, the stiffness matrix in global coordinate is found as:

$$[\mathrm{k}] = [T]^T[k'][\mathrm{T}] \qquad (5)$$

$$[k] = \frac{EA}{L}\begin{bmatrix} l^2 & lm & ln & -l^2 & -lm & -ln \\ lm & m^2 & mn & -lm & -m^2 & -mn \\ ln & mn & n^2 & -ln & -mn & -n^2 \\ -l^2 & -lm & -ln & l^2 & lm & ln \\ -lm & -m^2 & -mn & lm & m^2 & mn \\ -ln & -mn & -n^2 & ln & mn & n^2 \end{bmatrix} \qquad (6)$$

If the load vector is defined in the local coordinate, it must be redefined in the global coordinate system using the following transformation.

$$\{f\} = [T]^T\{f'\} \qquad (7)$$

Therefore, the stress of the element in the global coordinate is found as:

$$\{\sigma\} = [E][B]\{q'\} = [E][B][T]\{q\} \qquad (8)$$

Static Analysis

The static analysis results, namely, the nodal displacements are obtained by solving the following equation.

$$[K][q] = \{f\} \qquad (9)$$

For find the displacement matrix, Eq. (9) was multiply inverse of global stiffness matrix. In the FEM code, inv (\) command in MATLAB is used do the operation.
Global stress of the element in the global coordinate is,

$$\{\sigma\} = [E][B]\{q'\} = [E][B][T]\{q\} \qquad (10)$$

Dynamic Analysis

To perform dynamic analysis, mass matrices, which were derived in previous section are required. The following equation is solved and natural frequencies are obtained.

$$[M][\ddot{q}] + [K][q] = \{0\} \tag{11}$$

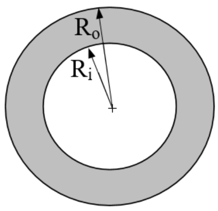
In the code, pinv(K) command in MATLAB which is based on Moore-Penrose pseudoinverse, is used to evaluate the inverse of matrices.

RESULTS AND DISCUSSIONS

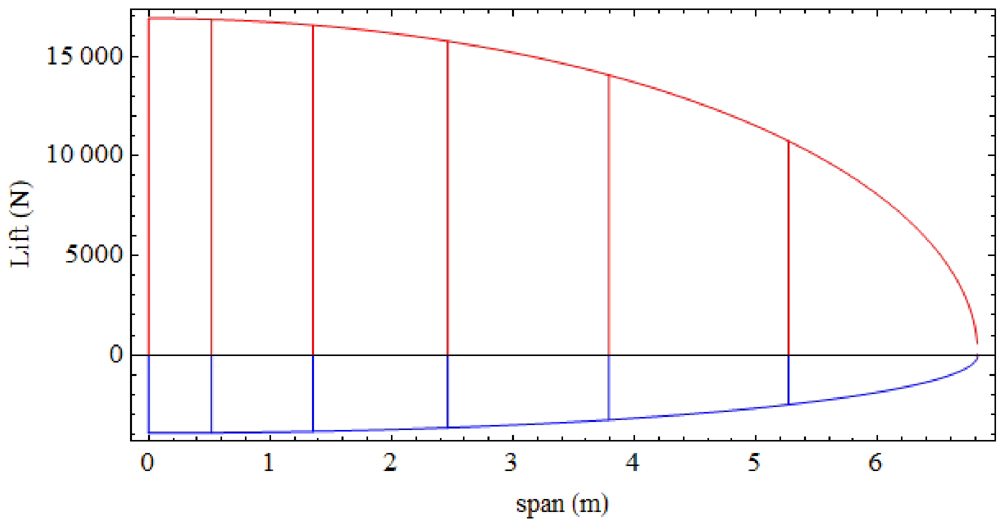
Wing Design

The wing structure is modeled using 1-dimensional truss elements. The cross-section of the elements are chosen to be hollow circle. A number of different cross-sections are considered and they are given in Table 2.

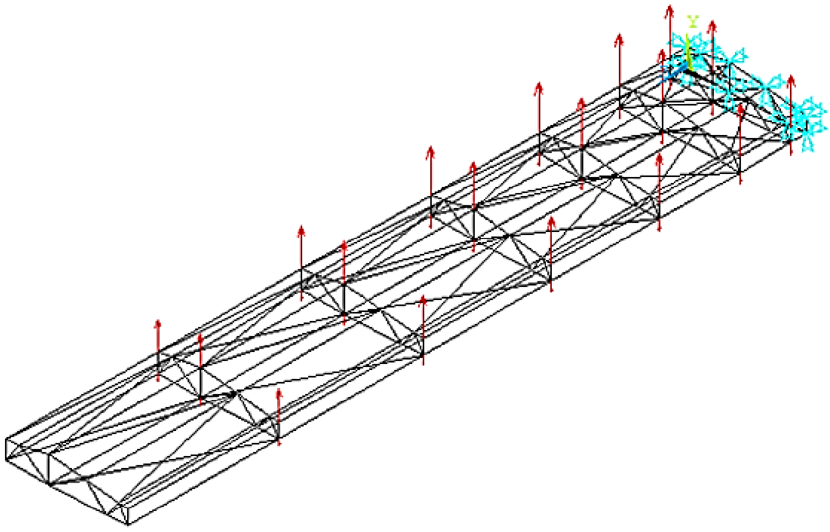
Table 2: Different cross-sections for truss elements in the wing design

	Area #1	**Area #2**	**Area #3**
	$R_0 = 5\ cm$	$R_0 = 5\ cm$	$R_0 = 10\ cm$
	$R_i = 3\ cm$	$R_i = 1\ cm$	$R_i = 0\ cm$
	$A = 12.57\ cm^2$	$A = 18.85\ cm^2$	$A = 78.54\ cm^2$

The aircraft weight is 34962.8 N, thus half wing has to carry the half weight of aircraft which is 17481.4 N. Multiplying this force by the load factor 4, the lift force is determined. More accurately, 130 % of the lift force is distributed to the upper surface in the upward direction, while 30 % is to the lower surface in downward section. Figure 6(a) shows the distribution of the lift force in spanwise direction and Figure 6(b) shows the applied boundary conditions, which is fixed free.



(a)



(b)

Figure 6: (a) Spanwise load distribution, blue and red lines represent lower and upper surfaces respectively, (b) Force and boundary conditions of the wing

Static Analysis Results

The nodal displacement results for the selected nodes and element stress results for the selected elements are listed in Table 2 and Table 3. The maximum displacement is measured as 357.84 mm at node 77.

Table 2: The nodal displacement results

Node #	ANSYS results			FEM Code results		
	δx	δy	δz	δx	δy	δz
11	-4.5282E-03	2.2665E+00	-1.1631E-02	-4.5282E-03	2.2665E+00	-1.1631E-02
12	-8.6953E-02	2.2707E+00	8.3712E-02	-8.6953E-02	2.2707E+00	8.3712E-02
13	-2.0791E-02	2.2709E+00	1.6563E-02	-2.0791E-02	2.2709E+00	1.6563E-02
14	-1.0804E-01	2.2815E+00	-2.1184E-01	-1.0804E-01	2.2815E+00	-2.1184E-01
15	-1.6274E-01	2.5725E+00	1.1159E-01	-1.6274E-01	2.5725E+00	1.1159E-01
16	3.1789E-02	2.8678E+00	4.6996E-02	3.1789E-02	2.8678E+00	4.6996E-02
17	3.8348E-02	2.9301E+00	0.0000E+00	3.8348E-02	2.9301E+00	-2.2691E-13
18	1.2169E-01	2.9269E+00	-1.2864E-01	1.2169E-01	2.9269E+00	-1.2864E-01
19	7.6101E-02	2.8316E+00	0.0000E+00	7.6101E-02	2.8316E+00	3.8958E-14
20	6.4665E-04	2.2879E+00	4.8308E-02	6.4665E-04	2.2879E+00	4.8308E-02
71	-2.5482E+00	2.9416E+02	3.6346E-02	-2.5482E+00	2.9416E+02	3.6346E-02
72	-1.2084E+01	2.9416E+02	3.6565E-01	-1.2084E+01	2.9416E+02	3.6565E-01
73	-1.3515E+01	3.0151E+02	-7.0360E-01	-1.3515E+01	3.0151E+02	-7.0360E-01
74	-1.6435E+01	3.1654E+02	-1.8654E+00	-1.6435E+01	3.1654E+02	-1.8654E+00
75	-1.4394E+01	3.3598E+02	4.7327E-01	-1.4394E+01	3.3598E+02	4.7327E-01
76	-1.2610E+01	3.5297E+02	2.8827E+00	-1.2610E+01	3.5297E+02	2.8827E+00
77	-1.2098E+01	3.5784E+02	0.0000E+00	-1.2098E+01	3.5784E+02	-7.3512E-13
78	-4.6738E+00	3.5784E+02	-9.7909E-01	-4.6738E+00	3.5784E+02	-9.7909E-01
79	-4.0594E+00	3.4683E+02	0.0000E+00	-4.0594E+00	3.4683E+02	1.7112E-12
80	-2.5245E+00	3.1654E+02	5.4446E-01	-2.5245E+00	3.1654E+02	5.4446E-01

The maximum tension stress is read as 345.4 MPa in the element 226, while the highest compression stress is read as 240.73 MPa in the element 173. The chosen material for the frame is AISI 1020 Steel [http://www.matweb.com/search/datasheet .aspx?bassnum=MS0001], which has a tensile strength of 420 MPa.

Table 3: The element stress results

Element #	ANSYS results	FEM Code results
18	4.4834E+00	4.4834E+00
22	9.7352E+01	9.7352E+01
38	-1.1912E+02	-1.1912E+02
49	-3.9651E+00	-3.9651E+00
56	4.0832E+00	4.0832E+00
60	2.1238E+01	2.1238E+01
71	-1.1696E+01	-1.1696E+01
74	4.5227E+01	4.5227E+01
82	-4.0696E+00	-4.0696E+00
162	-1.5510E+02	-1.5510E+02
173	-2.4073E+02	-2.4073E+02
185	1.0277E+02	1.0277E+02
207	1.2681E+02	1.2681E+02
224	-1.6643E+02	-1.6643E+02
226	3.4544E+02	3.4544E+02
235	-5.3716E+01	-5.3716E+01
240	-1.9619E+01	-1.9619E+01
245	5.3708E+01	5.3708E+01

According to the displacement and stress results, the wing design that yields an optimum material distribution and strength has been established.

Dynamic Analysis Results

The eigenfrequency results are listed in Table 4 for 10 mode shape.

Table 4: Eigen-frequency results

Mode #	Frequency	
	ANSYS results	FE Code results
1	3.6495E-02	3.6495E-02
2	1.2781E-01	1.2781E-01
3	1.6279E-01	1.6279E-01
4	2.3641E-01	2.3641E-01
5	3.7972E-01	3.7972E-01
6	4.5100E-01	4.5100E-01
7	4.7281E-01	4.7281E-01
8	6.5612E-01	6.5612E-01
9	8.8283E-01	8.8283E-01
10	9.7557E-01	9.7557E-01

REFERENCES

Lambert, M. (1990) *Jane's All the World's Aircraft, 1990/1991*, Jane's Information Group, Incorporated.

Logan, D,L. (2007) *A First Course in the Finite Element Method*, CL Engineering.

Sigmund, O. (2001) A 99 Line Topology Optimization Code Written in MATLAB, Educational Article, Springer-Verlag.

Walker, D. Liu, D. and Jennings, A. (2015) Topology Optimization of an Aircraft Wing, 56th AIAA Structural Dynamics and Materials Conference.

http://www.matweb.com/search/datasheet.aspx?bassnum=MS0001