Simulation research of the strength of an engine mount in an aircraft piston diesel engine

P Magryta and K Pietrykowski
Lublin University of Technology, Faculty of Mechanical Engineering, Lublin 20-618, Nadbystrzycka 36/710, Poland
p.magryta@pollub.pl, k.pietrykowski@pollub.pl

Abstract. The article presents strength simulations of a mount for mounting the test engine. Mounted on a stationary test stand, this mount consists of external fixings, fixings to stabilize the engine and tubular elements as a truss. These tubular elements are pipes made of seamless black steel. The material of the truss is S235JR steel. The article examines three different versions of the mount: mount no. 1 - initial mount, mount no. 2 - mount after a modification of pipe arrangement, mount no. 3 - mount after a modification of pipe wall thickness. For each version of the mount and subsequent calculation steps, the same boundary conditions and results legend were assumed. All calculations were made in Catia v5 in the Generative Structure Analysis module. To reflect the conditions prevailing during the engine operation on the test bench, the following conditions as mount load were adopted: gravity from the engine mass as 1000 N; engine thrust as 5000 N, and engine torque as 227 Nm. First, the model was pre-calculated to check the influence of mesh size on the obtained results. 2 mm parabolic tetrahedral elements were used in a computational grid. All subsequent steps of the mount modification showed a positive effect of reducing the maximum stress values or their mitigation as dispersion over a larger area. The changes made it possible to eliminate potentially dangerous areas of stress accumulation points. The material used has a strength several times greater than the stresses occurring in the tested elements. It was found that no further modifications to the mount are required and it is possible to use the created geometry on the test stand.

1. Introduction
Internal combustion engine testing is conducted on dynamometer test benches. To measure engine performance such as power or torque, engine load is necessary. Usually, load is a special brake simulating the resistance of the power receiver, i.e. vehicle wheels or an aircraft propeller. When testing aircraft engines, variable-pitch propellers are often used instead of a brake. These allow the engine to be loaded within the range of power and rotational speed encountered during flight. This method is also used in marine applications [1]. Additionally, the use of a propeller designed for future use with a given engine allows for analysis of the performance of the entire powerplant. One of the most important issues is fuel consumption [2].

Propeller-loaded aircraft engines are mounted on stands in a swinging manner. This allows measurement of reaction torque occurring at engine supports which is equal to torque generated by the engine under test. To mount the engine on the test stand, it is necessary to construct a mount to connect the engine to the test stand. Such mounts are often similar to designs used in aircraft. Besides strength, a correctly designed mount should have adequate stiffness. For complex
structures, these characteristics are best investigated earlier with a relevant type of software that is used at every stage of the engine design process. Starting from the conceptual phase, these are dimensional systems such as AVL Boost [3-5], CFD [6] or the finite element method (FEM) [7,8]. The article [9] describes a method of designing an engine mount for use in a small sports aircraft of the Zlin 26 series, based on the PTC ProE Mechanica software. The report [10] describes a FEM analysis, using the Cosmos-M software, of the engine mount for an Apollo Canard fixed-wing aircraft to verify adequate strength for all operating conditions.

The design process allows selecting the potentially best version or making necessary corrections even before the actual design is performed. The same process can be applied not only to engine elements but whole flying machines or their parts [11]. Also, elements used in the automotive design (engine mount) are simulated using FEM methods [12-15]. This is important because damage to the mount during testing can lead to engine failure, which significantly affects cost and time of testing. Additionally, it is possible to test vibration incurred by engine performance on the engine mount [16-18].

The paper discusses the simulation investigations of the strength of several versions of the suspension mount of the PZL-100 engine demonstrator. The target mount was mounted on a stationary test stand-in WSK "PZL-Kalisz" S.A. and the first geometrical version of the mount was delivered by this company. The mount has mountings screwed to the airmount, engine mountings (4 points) and elements made from closed profiles of circular cross-section. These elements are steel seamless tubes with an outside diameter of 1.2" (21.3 mm) and a wall thickness of 2.6 mm. The mount material is S235JR steel. The material data used in the simulation studies for those elements is presented below. Literature material data for this type of steel shows that its strength is $R_{e, min} = 235$ MPa and $R_m$ ranges from 360 to 510 MPa [19]. These data are defined by the PN-EN 10025-2:2019-11 [20].

### Table 1. Material data of S253JR steel.

| Property           | Value       |
|--------------------|-------------|
| Young's modulus    | 210 GPa     |
| Poisson's ratio    | 0.3         |
| Density            | 7800 kg/m³  |

This article presents the results of strength tests of three different mount versions:
- mount version no. 1 – base mount manufactured by WSK "PZL-Kalisz" S.A.,
- mount version no 2 – with a modified profile arrangement,
- mount version no. 3 – with a modified profile wall thickness.

The exact modifications are described in the following subchapters and all their views are shown in the article summary. The same boundary conditions and scale range of the results were assumed for each mount version and subsequent calculation steps, which enables a direct comparison of all versions. All calculations were performed in Catia v5 and the Generative Structure Analysis module.

### 2. Research object and boundary conditions

Figure 1 shows the clamping of the mount in a simulation environment. This figure shows the surfaces that were given the Clamp function, which immobilized them (all degrees of freedom taken away, rigid support).
The following conditions were used as mount loading to reflect the conditions during engine operation on the test bench:

- the gravity force from the engine mass: 1000 N, an engine mass of 100 kg estimated from the developed geometric models,
- the propeller thrust force as 5000 N, this value corresponds to the propeller thrust at maximum engine power,
- engine torque as 227 Nm, this value corresponds to the reaction torque of the propeller.

All these conditions were specified at the geometric centre of gravity of the engine. For this purpose, an additional element, i.e. a cubic lump located at the centre of gravity was used. This lump was then connected to the motor mount surfaces in a non-deformable manner using the Rigid Virtual Part function, as shown in Figure 2.
Figure 3 shows where the resultant of gravity and thrust and the torsional moment are applied.

![Figure 3. Location of gravity and thrust and the torsional moment.](image)

The model built this way was then subjected to preliminary calculations to check the effect of mesh size on the results obtained. Parabolic tetrahedral type elements were used in the computational mesh. The calculations were started with a single element of 5 mm reduced to 4, 3 and 2 mm. The subsequent reduction in element size did not affect the calculation results, so it was decided to use a mesh element of 2 mm.

3. Results

3.1. Version no. 1

Figure 4 shows the results in the form of stress maps for mount version no. 1. As can be seen, the highest stress values occur in the profiles located in the upper part of the mount. These are the elements most loaded with the tensile force. The maximum value of stress for these two elements is within the range of 20-25 MPa. These values are about 10 times lower than the allowable stresses for the accepted material. It should be noted, however, that the static loads were analyzed in the study, and due to the cyclic operation of the internal combustion engine, the mount will also be subjected to variable loads. The lower sections of the clamping mount have stress values in the range of 5-7.5 MPa. The other elements are blue, which means that the stresses in them are low (below 2.5 MPa). An uneven stress distribution can also be seen if we compare the left and right sides of the mount. This is mainly due to the action of the torsional moment. However, this unevenness is not significant, so the next steps did not need to change the design of the left side to the right side, which would hinder its assembly and making the mount.
3.2. *Version no. 2*

The second version of the engine suspension mount differed from the first one due to the relocation of four profiles located on the sides of the mount (two on each side). It was decided to mount these profiles as close to the engine mounts as possible. At the same time, it was checked whether such a change of mount geometry would not be incompatible with the designed engine. The change of mount geometry is shown in Figure 5.

**Figure 4.** Stress map for mount version no. 1.
Figure 5. Change in the geometry of mount version no. 2 (right) compared to mount version no. 1 (left).

Figure 6 shows the results in the form of stress maps for mount version no. 2. The calculations of the modified structure showed uniform stress distribution in the upper mount profiles. The maximum values remained the same, but their distribution became more uniform, which positively affects the strength and durability of the elements. This situation is similar to the lower mount elements. It should be noted that it is impossible to move the tubes further towards the engine mount due to a possible collision with the engine elements on the one side. However, as mentioned earlier, making an asymmetrical mount will cause difficulties when making the mount.
3.3. Version no. 3
Another modification was to increase the wall thickness of the two components highlighted in orange in Figure 7. These components are most heavily loaded. It was decided to keep their outer diameter so as not to disturb the overall geometry of the mount but increase their wall thickness to 4.2 mm. These tubes are depicted in Figure 7.

Figure 6. Stress map for mount version no. 2.

Figure 7. Profiles with the increased wall thickness.
Figure 8 shows the results in the form of stress maps for mount version no. 3.

As can be seen, increasing the wall thickness of the selected tubes significantly reduced the maximum stress values to about 12-12.5 MPa, which is a small value compared to the maximum stress limits for the material used. The geometrical changes made in mount versions 2 and 3, i.e. a change of wall thickness increased the strength of the mount structure.

Additionally, to visualize how the mount deforms, a visualization of the deformation of mount version no. 3 is shown in Figure 9. The visualization shows the deformation multiplied by a factor of 3000. The correct deformation values in mm are given in the legend. As seen, the maximum...
deformations are less than 0.04 mm, which allows us to conclude that no further modifications to the mount are necessary.

**Figure 9.** Visualization of the deformation of mount version no. 3.

### 4. Summary

The research was to optimize the engine frame structure to reduce stress. The first version of the engine frame showed the highest stress values of 20-25 MPa in the profiles in the upper part of the frame element. In the second version, the attachment points for the non-tubular elements were shifted and the result was that maximum values remained on the same level, but their distribution became more uniform. In the third version, the modification was to increase the wall thickness of the two most heavily...
loaded components. The wall thickness increased from 2.6 mm to 4.2 mm. The modification resulted in a significant reduction of the maximum stress values to about 12-12.5 MPa.

The subsequent modification steps of the frame geometry resulted in a reduction of the maximum stress values and also enabled to eliminate potentially dangerous points of stress concentration. The applied material has a strength several times higher than the stresses in the tested elements. In conclusion, further modifications to the frame are not necessary and the created geometry can be used on the test stand.

Acknowledgements
This work has been realized in cooperation with The Construction Office of WSK "PZL-KALISZ" S.A." and is part of Grant Agreement No. POIR.01.02.00-00-0002/15 financed by the Polish National Centre for Research and Development.

References
[1] Yang Z and Wen H and YangX and Gorbov V and Mitienkova V and Serbin S Matching Characteristic of Hull, Eninge and Propeller Marine Power Plant pp 183-202
[2] Tadros M and Vettor R and Ventura M and Soares C G 2021 Coupled Engine-Propeller Selection Procedure to Minimize Fuel Consumption at a Specified Speed Journal of Marine Science and Engineering 2021, 9, 59. https://doi.org/10.3390/jmse9010059
[3] Grabowski Ł and Pietyrkowski K and Karpiński P 2019 The zero-dimensional model of the scavenging process in the opposed-piston two-stroke aircraft diesel engine Propulsion and Power Research, 8(4). ISSN 2212-540X, https://doi.org/10.1016/j.jppr.2019.11.003
[4] Magryta P and Pietyrkowski K and Gęca M 2018 Thermodynamic Model of the ASZ-62IR Radial Aircraft Engine Transactions on Aerospace Research, 2018(1), https://doi.org/10.2478/tar-2018-0004
[5] Czyż Z and Grabowski Ł and Pietyrkowski K and Czarnigowski J and Porzak M 2018 Measurement of flight parameters in terms of toxic emissions of the aircraft radial engine ASz62-IR Measurement, 113. ISSN 0263-2241, doi.org/10.1016/j.measurement.2017.08.035
[6] Biały M and Pietyrkowski K and Tulwin T and Magryta P 2017 CFD numerical simulation of the indirect cooling system of an internal combustion engine Combustion Engines 170(3):8-18, https://doi.org/10.19206/CE-2017-302
[7] Pietyrkowski K and Magryta P and Skiba K 2019 Finite element analysis of a composite piston for a diesel aircraft engine Combustion Engines 179(4):107-111, https://doi.org/10.19206/CE-2019-417
[8] Magryta P. and Michał G 2019 FEM analysis of piston for aircraft two stroke diesel engine MATEC Web of Conferences 252, 07004 https://doi.org/10.1051/matecconf/201925207004
[9] Jurek M and Judičák J 2013 Design and analysis of engine mount for the use of modern-design engine in small sport plane Zlin 26 Series Acta Avionica. 15(27)
[10] 2003 Engine mount analysis report. Report No ER-01002
[11] Skiba K 2019 Designing and FEM simulation of the helicopter rotor and hub IOP Conference Series: Materials Science and Engineering, Volume 710
[12] Maski s and Basavaraj y 2020 Finite element analysis of engine mounting bracket by considering pretension effect and service load IJRET: International Journal of Research in Engineering and Technology eISSN: 2319-1163 | pISSN: 2321-7308
[13] Singh D and Kumar R 2019 Stress analysis of engine mounting assembly of a three wheeler IOP Conf. Series: Materials Science and Engineering 691 (2019) 012064 doi:10.1088/1757-899X/691/1/012064
[14] Alkhatib F 2013 Techniques for Engine Mount Modeling and Optimization University of Wisconsin Milwaukee UWM Digital Commons PhD thesis
[15] Adkine A S and Overikar G P and Surwase S S 2017 Modal Analysis of Engine Supporting Bracket using Finite Element Analysis International Journal of Advanced Engineering Research and Science (IJAERS) 4(3), Mar-2017 https://dx.doi.org/10.22161/ijaers.4.3.9

[16] Ramos F M D 2008 Vibration analysis of an engine mount Departamento de Engenharia Mecânica Instituto Superior Técnico, Universidade Técnica de Lisboa

[17] Ravi Teja Reddy M and Jayakumar V and Santhoshkumar D and Muniappan A 2018 Vibration Testing of Novel Engine Mount Technical Note Int. J. Vehicle Structures & Systems, 10(6), 417-419 doi:10.4273/ijvss.10.6.09

[18] Lu-Ean Ooi and Zaidi Mohd Ripin 2016 Optimization of an engine mounting system with consideration of frequency dependent stiffness and loss factor Journal of Vibration and Control2016, 22(10) 2406–2419 doi: 10.1177/1077546314547532

[19] Rdzawski Z and Pacyna J and Głuchowski W and Dąbrowski R and Rożniata E and Sobota J 2018 Własności i struktura stali S235JR po statycznym i dynamicznym przeginaniu i prostowaniu na zimno Obróbka Plastyczna Metali vol. XXIX No 2

[20] PN-EN 10025-2:2019-11 Wyroby walcowane na gorąco ze stali konstrukcyjnych - Część 2: Warunki techniczne dostawy stali konstrukcyjnych niestopowych.