1 Introduction

1.1 The problem of aerodynamic drag

Dlaczego badamy oraz wzór na opór aerodynamiczny. Że bazujemy na modern exterior balistics i jakieś inne z literatury bo to łądnie brzmi.

1.2 Methodology of the present work

For simulations, two programs were chosen to compare the results. The first program, Solidworks Flow Simulation, was used for both CFDs and model preparation. The second program utilized was Ansys Fluent.

Initially, the models were prepared in Solidworks and subsequently exported to .step (214) file format for importation into Ansys. Within Ansys, Fluent with Meshing was used to prepare the mesh, followed by the execution of simulations. Solidworks Flow Simulation was also employed for mesh preparation and simulation execution, enabling subsequent comparison with results obtained from Ansys Fluent.

Parametric studies/sets were conducted for all models, encompassing nine different velocities ranging from 0.1 to 1.0. Subsequently, resulting graphs depicting the drag coefficient versus Mach number were analyzed and compared.

1.3 Tested models

R6-Endcone, R6-No-Endcone, PrawieR5

2 Upgdated R5 model

- Domena i mesh
- Kolorki dla 0.2, 0.5, 0.8
- Wykres CD

3 R6 Endcone

3.1 Solidworks

- Domena i mesh
- Kolorki dla 0.2, 0.5, 0.8

3.2 Ansys Fluent with meshing

- Domena i mesh
- Kolorki dla 0.2, 0.5, 0.8

Wykresy obu na koniec zestawić.

4 R6 No Endcone

4.1 Solidworks

- Domena i mesh
- Kolorki dla 0.2, 0.5, 0.8

4.2 Ansys Fluent with meshing

- Domena i mesh
- Kolorki dla 0.2, 0.5, 0.8

Wykresy zestawić.

5 Results and discussion

- Wykresy CD dla wszystkich modeli
- Opis porównania i zestawienie z literaturą