TRANSACTIONS OF THE INSTITUTE OF AVIATION
No. 1(242), pp. 7-22, Warsaw 2016
DOI: 10.5604/05096669.1202199

ISSN 0509-6669
eISSN 2300-5408

CFD ANALYSIS OF ROTOR WAKE INFLUENCE ON ROOFTOP HELIPAD OPERATIONS SAFETY

Adam Dziubiński
Institute of Aviation, al. Krakowska 110/114, 02-256 Warsaw, Poland, adziubinski@ilot.edu.pl

Abstract

In the following work a real design of helipad and two example helicopters data are used to illustrate the methodology of using CFD in prediction of helipad operation safety. The analysis is limited to cases of hover, where the influence of the main rotor wake is dominant. Both safety of helicopter and the on-deck helipad crew are taken into account, so the flow velocity on the helipad surface and pressure distribution are obtained as well as the wake – induced flow character near the well-shaped part of a building. The flow is solved using commercial code solving RANS equations with finite volume method. Spalart Allmaras turbulence model is used, since mainly the turbulent flow occurs on rotor wake, and flow is defined around blunt bodies. This method is proven to be useful for rotational fluid cases, and trimmed rotor modeling for hover near building cases. The results were used in design of the above mentioned helipad.

Keywords: CFD, helicopter safety, helipad, rotor wake.

1. INTRODUCTION

The main difference between helicopter and fixed wing aircraft operation is, that it operates not only from flat airfields, but also from helipads in towns or mountains. Shape of the terrain and obstacles surrounding the helicopter deflect the flow around it, sometimes in unpredictable for pilots manner. That could cause a lot of trouble until specific phenomena is recognized and the pilots learn to avoid certain flight conditions over specific locations.

In the following work, a test case of real design of helipad is used as an example to show the wake influence on helipad and surrounding buildings. This test case fits very well for its purposes as an example of neighboring building size and shape influence on wake distribution near helipad. This case also fits purposes of a situation when the hover can be dangerous, because of flow interaction of main rotor and surrounding object, which could cause lack of power sufficient to keep hover and in consequence lead to the emergency landing.

1.1. Scope of work

This work is an extension in area of research on the Institute of Aviation to the topics related to the safety in construction and medical care, which is coincident with an IoA policy of work [13]. The real geometry used in this research was the COPERNICUS Hospital Center facility in Gdansk, equipped with planned helipad, placed on a tower between the buildings (Fig. 1 right). The surrounding buildings were also taken into account to properly model the flow phenomena around the helipad. Using photogrammetric method and the aerial photography (Fig. 1 left), the geometry of the facility with the surroundings was reconstructed with accuracy suitable for this CFD simulation.

The calculation of rotor wake influence has been divided into following topics:

- Analysis of rotor wake influence on helipad.
- Comparison of two types of helicopters: civil medical transport (EC-135) and military medevac (W-3RM).
- Analysis of wake influence on surrounding objects.
- Research for potentially dangerous locations of helicopter in hover taking into account aerodynamic interference.

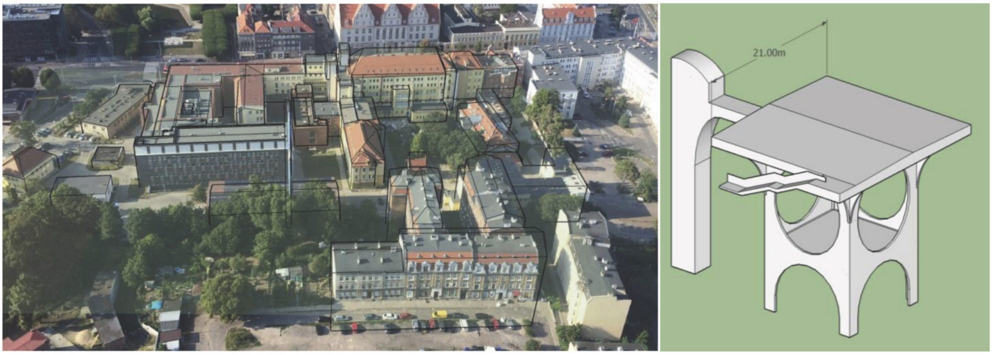

Fig. 1. Comparison of 3d geometry and aerial photography (left) and geometry of planned helipad tower (right) [5]

Eurocopter EC-135 is an example of civil medical transport helicopter. PZL Swidnik W-3RM Anakonda is a military transport helicopter (Fig. 2 left). The first of them is used as a monotype in the State Medical Transport Aviation, and the other is a standard medevac helicopter in the Polish Navy. Checking those two types was crucial to obtain conditions of helipad usage.

The main rotor wake has been, in the following simulation, represented by actuator disc, on which a constant pressure jump has been assumed. In this way the model represents a trimmed rotor in hover. As a baseline, the Eurocopter EC-135 rotor has been used, for a helicopter with mass equal to maximum takeoff mass (MTOW) [3]. The calculations have been repeated for one position and increased disc diameter corresponding to PZL W-3RM ”Anakonda” helicopter. On all simulations the fuselage has been omitted, which corresponds to the worst case scenario for helipad surface loading.

The pressure jump has been calculated using the following formula:

$$dP = \frac{m \cdot g}{S} = \frac{MTOW \cdot g}{\pi r^2}$$

where:
g – standard gravitational acceleration on Earth [m/s^2],
MTOW – maximum takeoff weight [kg],
r – rotor radius [m].

The results have been shown in Tab. 1. It is worth mentioning, that since W-3 RM is twice as heavy as EC-135, the average pressure jump on both rotors is similar, and even lower on heavier PZL W-3RM Anakonda.

Tab. 1. Comparison of helicopter data [3, 9]

	EC-135	PZL W-3RM Anakonda	
MTOW	2910	6400	kg
r	5.1	7.85	m
S	81.7	193.6	m^2
dP	349.36	324.31	Pa

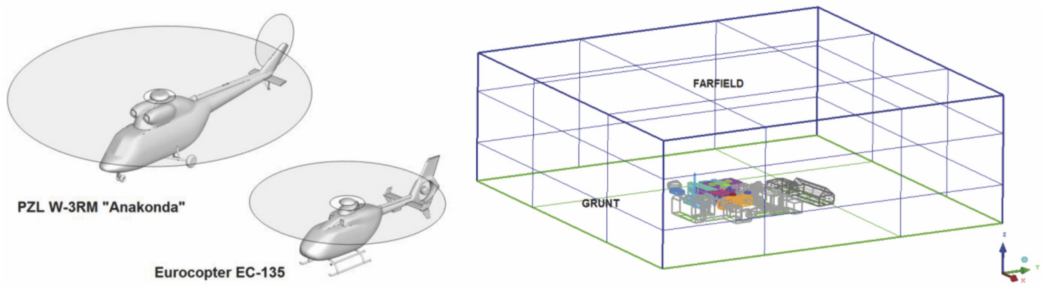

Fig. 2. Size comparison between EC-135 and W-3RM [10, 11] (left) and overview on computational domain (right) (Dziubiński, 2015)

2. RESEARCH METHOD

The flow simulations have been computed using Reynolds Averaged Navier Stokes (RANS) equation solver based on finite volumes method. The main set of equation has been closed with Spalart Allmaras turbulence model equations, known as standard in external flow simulations but assuming a turbulent flow [1]. The choice of one-equaton turbulence model, instead one of the two-equation eddy-viscosity models, has been made, because no buoyancy or phase-dispersion effects are to be modeled. There, such models have a great advantage over one-equation ones. The robustness of this method has been also a factor, since, as it is stated in work of Zhang et al. [12], ”the S-A model is local so that the solution at one point is independent of the solutions at neighboring cells and thus compatible with grids of any structure”. So this is a proper model for the solution of rotor wake flow, with unstructured grid, constrained with blunt geometry of buildings. Basing on the author’s experience, this model is also less prone to cause the flow solution divergence, when strongly separated flows are calculated.

Much better in prediction of separated flow, Large Eddy Simulation (LES) model, is so demanding in terms of mesh density and user experience, that it was impossible to use it in this specific case, but this could be one of the future study directions.

2.1. Geometry zones

The geometry model of facility and helipad has been divided into smaller zones in order to set proper boundary conditions. Around this geometry a computational domain has been generated. The domain was a rectangular shape cuboid size of 600 by 600 m on base rectangle and 200 m of height. The top and sides of the domain boundary conditions (FARFIELD at Fig. 2) have been set as Pressure Far Field. Bottom surface of domain (GRUNT at Fig. 2) and building walls have been set as impenetrable by flow (wall type of boundary condition). In Fig. 3 the division into zones has been shown.

Fig. 3 and 4 show also the surfaces that are meant to serve as switchable actuator discs (FAN). Switching off the actuator disc means that an „INTERIOR" boundary condition is set, so the surface becomes nonexistent to a flow. It is assumed, that only one of the available actuator discs is on during the simulation. Exception is a W-3RM rotor, which consists of two zones: internal disc FAN_C representing smaller EC-135 rotor and external ring FAN_C_ANAKONDA representing additional surface of W-3RM main rotor. In Fig. 3 (left) the description of that geometry is shown. Each configuration assumes, that a pressure jump on active parts of fan surface will be corresponding to mean value of pressure jump for chosen helicopter.

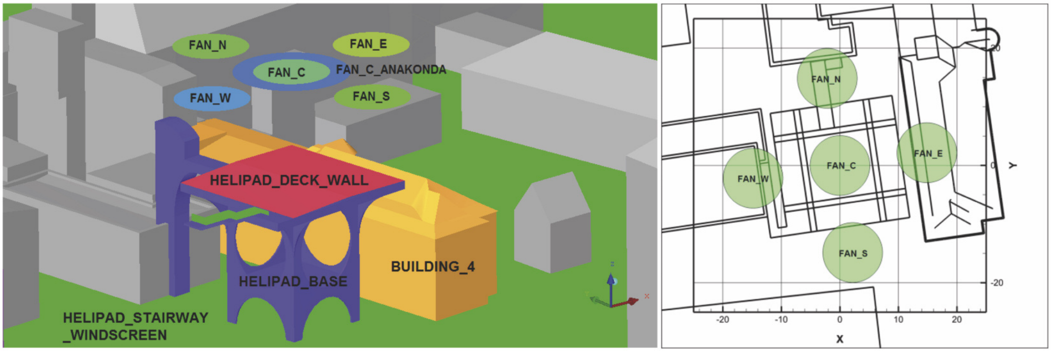

Fig. 3. Zone names for hover over the helipad cases, side view with part of buildings omitted (left) and top view of fan positions (right) (Dziubiński, 2015)

Two groups of actuator disc surfaces have been created. One contains test cases for helipad loading and the other simulates a helicopter hovering around well shaped building parts, which have been claimed as the dangerous areas in works of Łusiak et al. (2009) [4] and Sobczak (2008) [6].

The test cases for helipad load caused by the main rotor wake have been selected to test the hover in the following situations:

- over the center of helipad: FAN_C (+ FAN_C_ANAKONDA optionally),
- where the neighbour building with flat roof is similar size to the helipad tower: FAN_W,
- where the neighbour building with flat roof is lower than the helipad tower: FAN_N,
- similarly to above case but the building has pitched (gable) roof: FAN_E,
- when the building with gable roof is smaller and distant: FAN_S.

Those cases practically cover all of the rooftop neighboring conditions, because usually no one designs the helipad near the building, that is higher than helipad surface, unless it is unavoided, as for example on sea ship (marine) helipads. But those helipads are difficult in operations and demanding a specific procedures and trained crews. The names of test cases have been set using geographical directions.

The test cases for dangerous places to hover (shown in Fig. 4 left) are as follows:

- Above junction of closed corridors – FAN_WELL_1.
- Above junction of parially open corridors – FAN_WELL_2.
- Above slipped roof near wall FAN_APR_2 – not used in CFD calculations.
- Above slipped roof in the corner FAN_APR_1 – not used in CFD calculations.

Additionally, the two cut-planes have been defined (Fig. 4 right) to adequately visualize the resulting flow caused by the actuator discs.

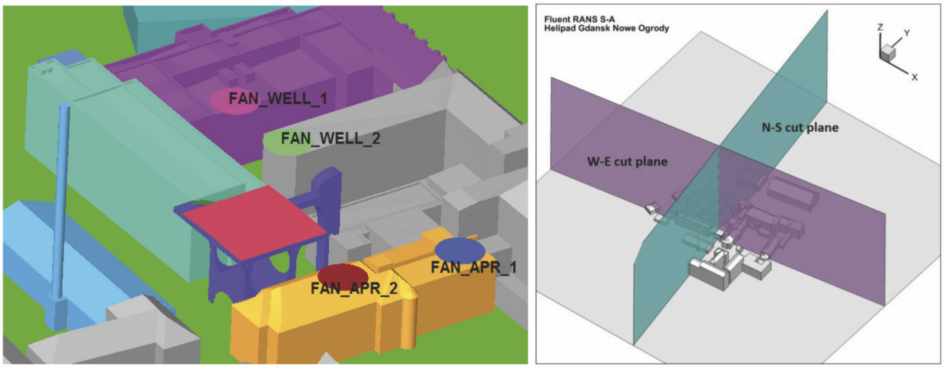

Fig. 4. Actuator discs surfaces for hover over the well shaped areas (left) and cut planes defined for hover over the helipad visualization (right) (Dziubiński, 2015)

2.2. Computational mesh

Around the helipad and buildings a computational mesh has been generated. An appropriate nonstructural mesh has been generated with density distribution set as it has been shown in Fig. 5, in order to properly model the flow around all hard edges of the geometry. Structural mesh has also proved to be efficient in studies of Świderski (2008) [7]. All the edge lines have the density set to divide space into smaller elements than the surrounding walls. The mesh is automatically smoothed so the difference of size between neighboring cells is below corresponding margin.

An increase of the mesh density has also been set on actuator discs, especially on edges of those surfaces. The actuator discs influence has been modeled as a pressure jump, so proper discretization (division into finite cells) in vicinity of those areas is crucial to obtain adequate results. The rule of thumb is to generate a dense mesh where a high spatial changes of flow parameters are expected. In this way the dissipation of phenomena because of insufficient discretization density is avoided.

The computational mesh, size of above 3 000 000 elements, was created using ICEM CFD software, using ”Robust (Octree)” method. Maximum element size was set as 30 m. The size of elements on the edges of helipad and close neighbor buildings was set to 0.2 m and on the walls of helipad a size of 0.5 m. On the surrounding buildings size of 2 m on the wall and 0.8 m on the edges has been set. At the edge of actuator disc a mesh size of 0.3 m has been set and the disc surface mesh density was set to 0.8 m. Prismatic boundary layer area have not been created since mainly the detached flows with clearly marked separation are predicted.

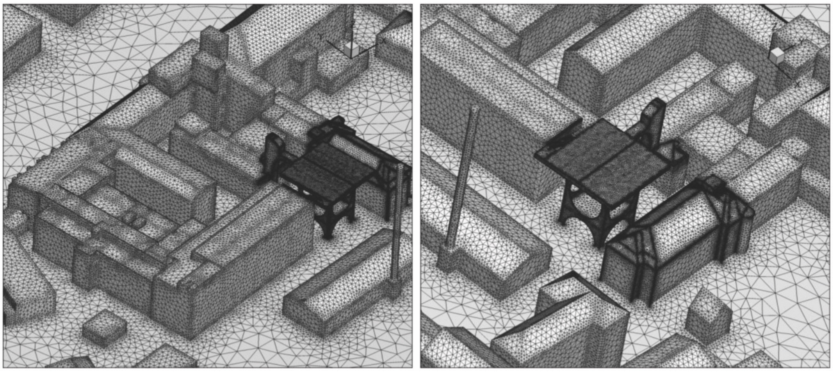

Fig. 5. Mesh density on surface of the model (Dziubiński, 2015)

2.3. Boundary conditions

All cases have been computed with accordance to ISA atmosphere model on sea level. That means, that a pressure have been set to 101325 Pa, Temperature to 288.15 K and air density to 1.225 kg/m^3. In simulation the following assumptions have been used:

- Stationary flow.
- Althrough the fluid is viscid, the specific mesh area representing the boundary layer has been ommitted, because of the object geometry, consisting so many hard edges, that separation is clearly marked, and a flow turbulisation in b.l. is not a factor.
- The pressure jump on actuator disc is averaged and corresponds to maximum takeoff weight, which is the worst case scenario.
- Ground surface is flat.
- Influence of tail rotor is neglected.
- Basic test case corrsponds to EC-135 helicopter hovering 15 m above a helipad surface.

Only the hover cases are in the scope of work. This kind of simulation, where no far field velocity is assumed, has a strong tendency to divergence. To avoid it, the boundary condition on pressure far field zone called FARFIELD (Fig. 2) around domain has been left to have zero magnitude of velocity, but the domain was initialized with non zero velocity values inside. Value of 1 m/s worked fine. The turbulence in the simulation has its only source inside the domain, so, on the FARFIELD, the default values of turbulent viscosity and dissipation ratio have been left.

On all the buildings and the ground surface, a ”wall” boundary condition. is assumed. Depending on which actuator disc is active, the other ones are set as ”interior” boundary condition. The active one is then modeled with ”fan” b.c. with constant pressure jump profile, set appropriately as in Tab. 1.

In other works, like Ruith [8], a more accurate rotor model was used, for the rotorcraft V-22 that is a multirotor type, where the interaction between two rotors and a helipad was essential to predict the stability of such design. Here only one, trimmed rotor is under consideration, so such detailed modeling is not necessary. The coordinate system used in following work centers in middle of the landing surface (above ground level), and has been oriented in coordination with geographic directions. The x axis is directed east, y axis is directed north and z is a vertical axis directed upward.

3. RESULTS

The results obtained in described above computations are shown below as the velocity and pressure maps on specified surfaces, defined as flow cross sections and wall surfaces. The near wall velocity has to be treated more as qualitative information than an exact value. This chapter has been divided into following parts: hover over the helipad, comparison between two sizes of helicopter and hover over well shaped (dangerous) areas.

3.1. Hover over the helipad

Fig. 6, 7 and 8 are containing distribution of static pressure and near wall velocity on the walls of the model. Surface pressure (Fig. 6 and 7 left) distribution is almost constant except areas, where the wake touches the ground or building below the rotor. Different situation appears, if the near wall velocity is analyzed.

When wake touches the helipad surface on the center (FAN_C), the stream is not constrained by the surrounding buildings, and stream flows around with almost constant intensity. When the rotor blows at the gap between building and helipad (FAN_W), the stream flows down to the ground, and then finds its way on corridors between buildings. This situation also constrains the demands for such helipad operations safety, because either people on the ground and on walkways to the helipad have to be secure and covered by sudden gusts, which could be dangerous and could cause them fall from high altitude.

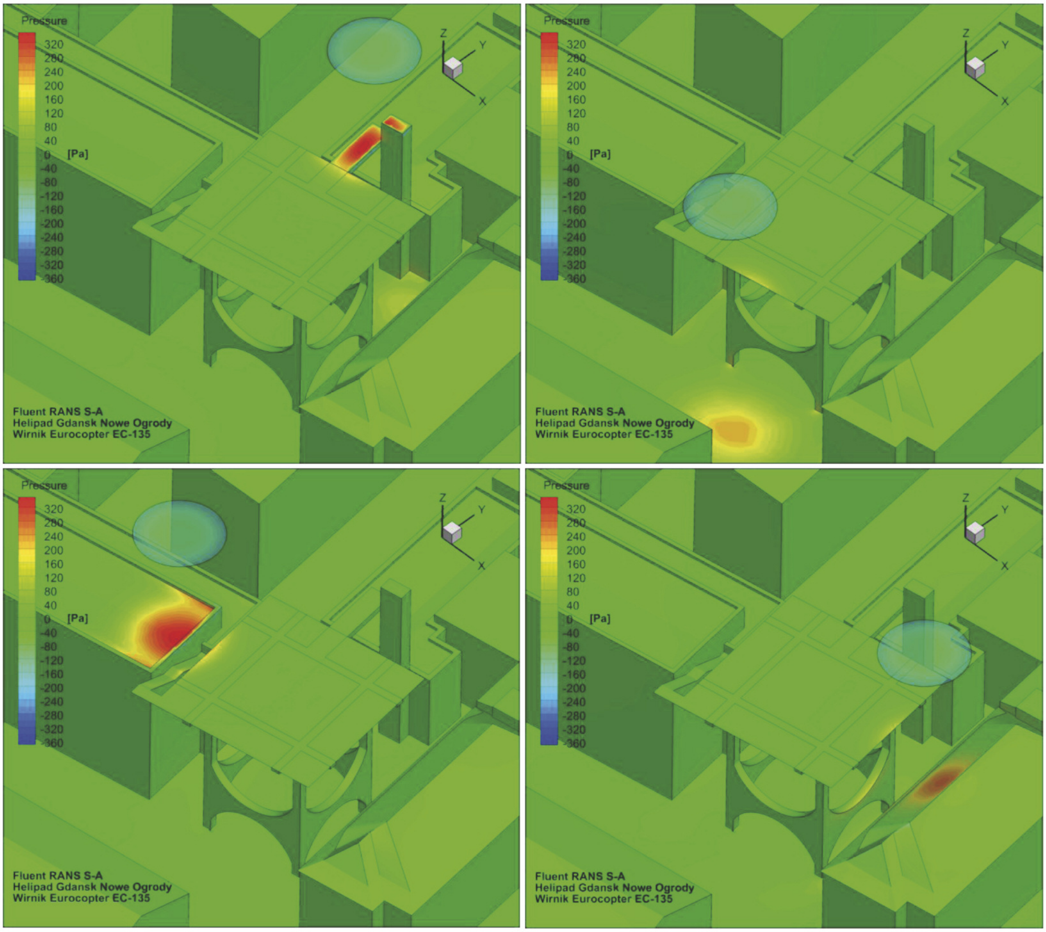

Fig. 6. Surface pressure for FAN_N (top right), FAN_S (top left), FAN_W (bottom left) and FAN_E (bottom right) cases (Dziubiński, 2015)

In case FAN_N, when helicopter hovers over the walkway to the elevator shaft, the man area of overpressure appears only on mentioned above walkway and the roof of a shaft. High near wall velocity, on the other hand, appears on the helipad surface (~15 m/s) and on the roof of nearest building and nearest corridors between buildings (~13 m/s).

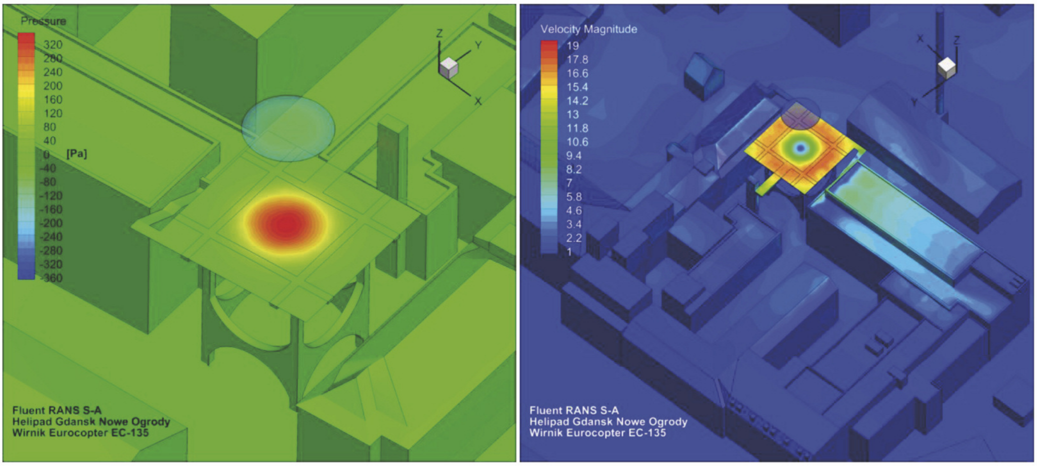

Fig. 7. Surface pressure (left) and near wall velocity (right) for FAN_C case (Dziubiński, 2015)

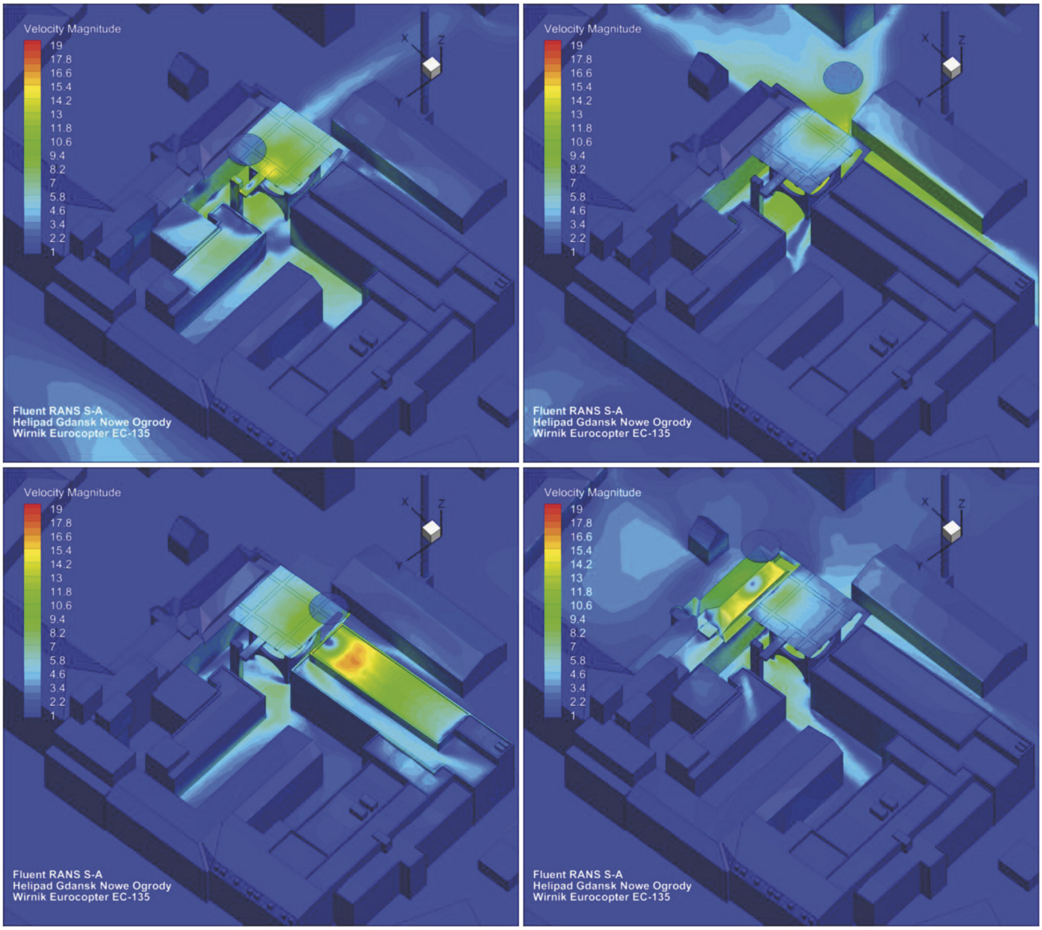

Fig. 8. Near wall velocity for FAN_N (top right), FAN_S (top left), FAN_W (bottom left) and FAN_E (bottom right) cases (Dziubiński, 2015)

When hovering over the pitched roof (FAN_E), the overpressure on the roof surface, about 320 Pa, is lower than the one caused by the shingles mass, which of course is much less than the force caused by the snow cover, 43 kg/m^2 = 422 Pa. Moreover, the force direction is coincident with usual load of the shingles, for which a most dangerous load is caused by underpressure. Since this building in real world is classified as a monument, there is a strong need to check for such dangers. The air velocity on the helipad is minimal, but a wake divided by the roof causes the higher velocity on the ground, but still it is no more than 30-40 km/h, which could be classified as safe conditions.

When helicopter hovers over the almost empty place in the case FAN_S, only surrounded by low building, a boiler house, an overpressure region can be observed mainly on the ground surface (size of 200 Pa). Low pressure jump, of about 120 Pa can be observed on helipad edge. In this case highest velocities appears on the bottom of the helipad tower, about 14 m/s. On the helipad surface it is only 10 m/s.

In order to show the flow inside fluid area, two cut planes (Fig. 4) have been defined in directions perpendicular to the walls of nearest high building, similar to geographic directions. Therefore these cut planes has been named as N-S (north-south, cutting through FAN_N, FAN_C and FAN_S cases) and W-E (west-east, cutting through FAN_N, FAN_C and FAN_S cases). In Fig. 9 and 10 a velocity magnitude distribution is shown as well as in Fig. 11 and 12 the static pressure distribution.

When the helicopter hovers over center of a helipad, the wake is divided sideways and loses its strength. The flow has tendency for descending down if the nearest roof is lower than the helipad surface, especially when the roof is pitched. The hover over any gap, on the other hand, causes the flow to be higher in magnitude and go below the helipad causing high velocities near the tower base, especially when the roof is, again, pitched. It is worth to mention that the flow returns by the gap on the opposing side than the helicopter hovers, which could also cause danger on ground crew, not expecting any air stream from this direction. Also the higher load on rainwater ducts, which usually are mounted below the roof edge, in this side is expected.

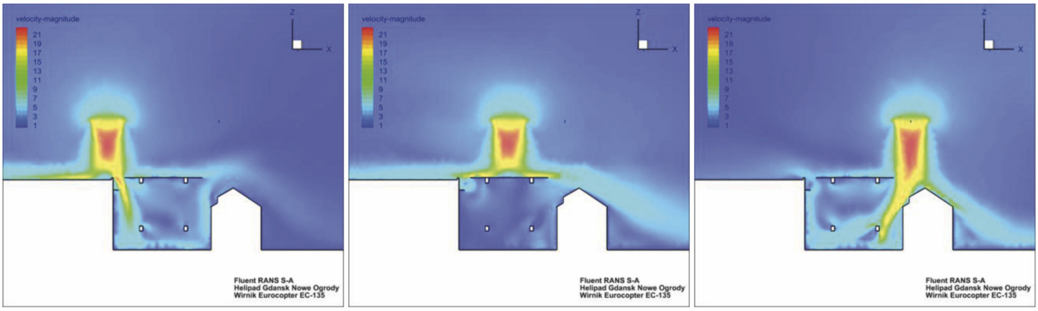

Fig. 9. Velocity magnitude distribution on the W-E cut plane, for cases FAN_W, FAN_C and FAN_E respectively (Dziubiński, 2015)

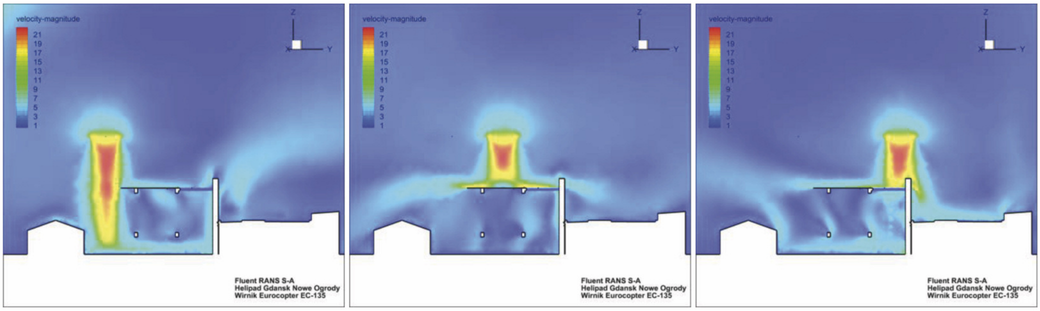

Fig. 10. Velocity magnitude distribution on the N-S cut plane, for cases FAN_N, FAN_C and FAN_S respectively (Dziubiński, 2015)

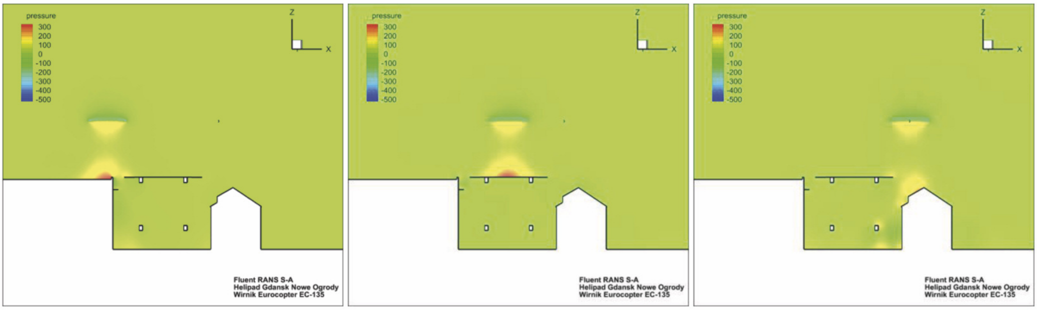

Fig. 11. Static pressure distribution on the W-E cut plane, for cases FAN_W, FAN_C and FAN_E respectively (Dziubiński, 2015)

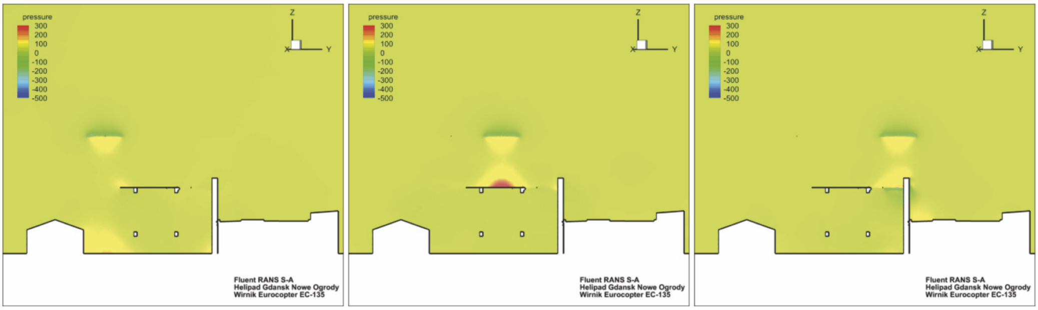

Fig. 12. Static pressure distribution on the N-S cut plane, for cases FAN_N, FAN_C and FAN_S respectively (Dziubiński, 2015)

Fig. 11 and 12 show the pressure distribution on described above cases. Again the pressure increase is visible on the wake, but it is worth mentioning, that its intensity depends strongly on distance of the obstacle and its wall inclination to flow.

3.2. Comparison of wake influence between EC-135 and W-3 RM Anakonda

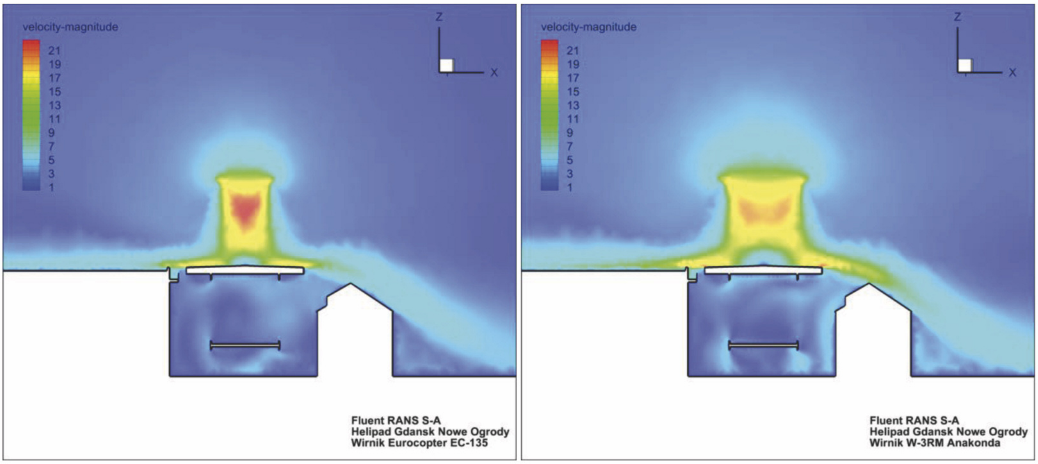

Fig. 13. Velocity magnitude comparison between Eurocopter EC-135 (right) and W-3RM Anakonda (left) in two cross sections (Dziubiński, 2015)

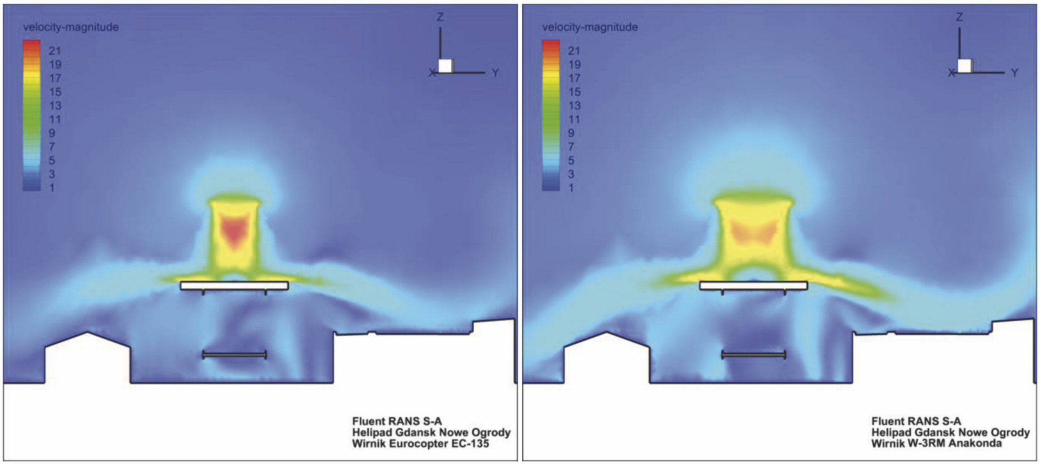

Fig. 13. continued. Velocity magnitude comparison between Eurocopter EC-135 (right) and W-3RM Anakonda (left) in two cross sections (Dziubiński, 2015)

In similar manner the comparison between two sizes of helicopter is shown, in Fig. 13 there is a velocity magnitude comparison as well as in Fig. 14 a pressure distribution on both cut planes. Both calculations have been done for helicopter hovering with rotor surface 15 meters above helideck. Because of geometric differences between Eurocopter EC-135 and W-3RM Anakonda rotors, two sizes of actuator disc have been used.

The results prove that the Anakonda is no more dangerous for nearest buildings than a EC-135, and that was the main concern, ability to use both helicopters (Polish Navy collaborates with civil hospitals) drastically increases safety of the region. Maximum underpressure probed on the monument building is around 50 Pa (5 kg/m^2) – when mass of the shingles is above 43 kg/m^2.

In Fig. 15 the difference between surface pressure distribution for both cases is shown. What is interesting, of course the size difference is obvious, but maximum pressure, size of 320 Pa, is similar for both cases, which means, that the bigger helicopter should not cause more surface degradation than a smaller one.

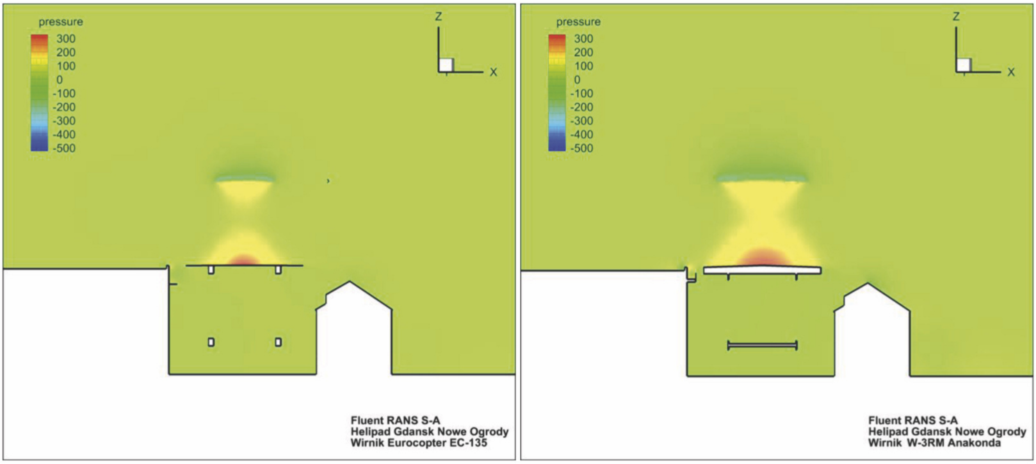

Fig. 14. Static pressure comparison between Eurocopter EC-135 (right) and W-3RM Anakonda (left) in two cross sections (Dziubiński, 2015)

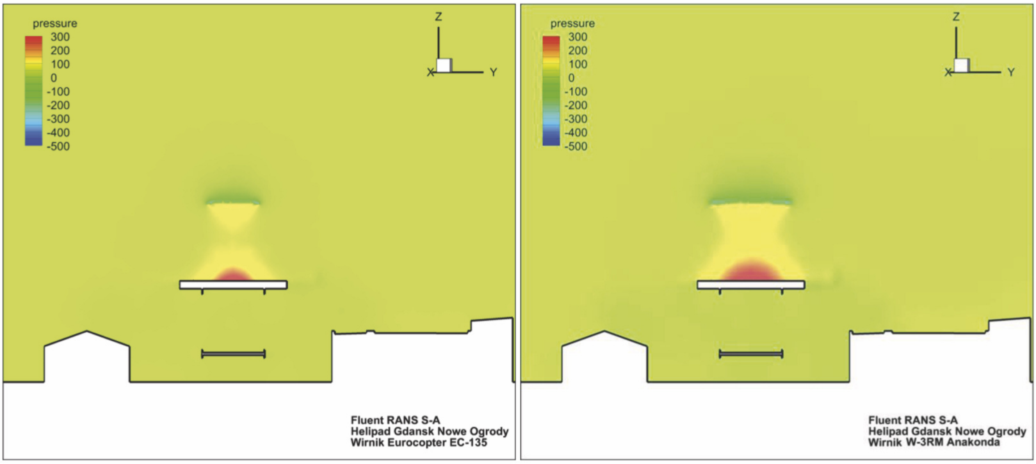

Fig. 14. continued. Static pressure comparison between Eurocopter EC-135 (right) and W-3RM Anakonda (left) in two cross sections (Dziubiński, 2015)

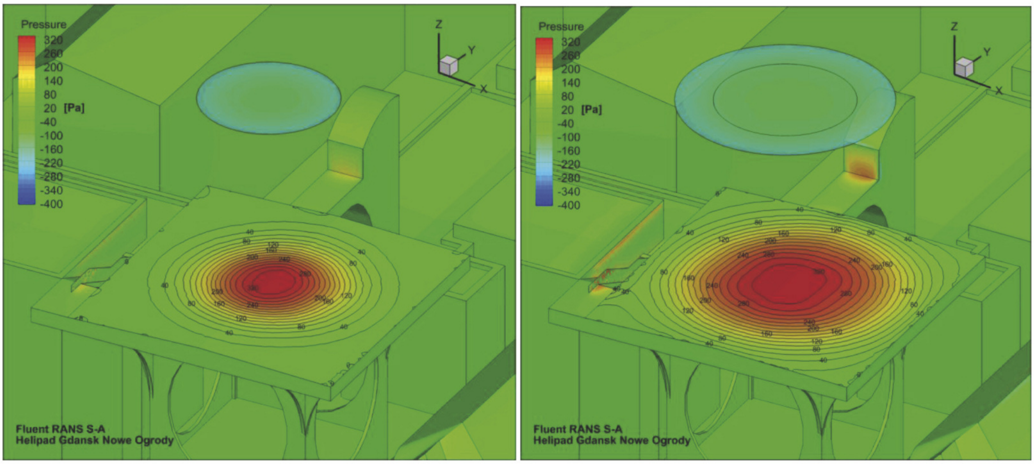

Fig. 15. Pressure load comparison between Eurocopter EC-135 (left) and W-3RM Anakonda (right) helicopters (Dziubiński, 2015)

3.3. Hover over well shaped (dangerous) areas

The two most dangerous areas of hover has been designated for CFD analysis, namely FAN_WELL_1 and FAN_WELL_2. Potentially dangerous appear the area surrounded by the walls, or well shaped cavity on the ground. When the helicopter hovers over such area, around main rotor appears the flow structure similar to the vortex ring state with all the consequences. The vehicle can be then out of power to move up from that hover, and forced to land on the bottom. The vortex appearing on the part of rotor is also dangerous, because then the helicopter can be pulled toward the side wall.

In Fig. (16) the results of simulation from Łusiak et al (2009) [4] and Dziubiński et. al. (2008) [2] have been shown, where such vortex structure appears, and on first of them the helicopter is deep inside such vortex ring, lacking the power to move up.

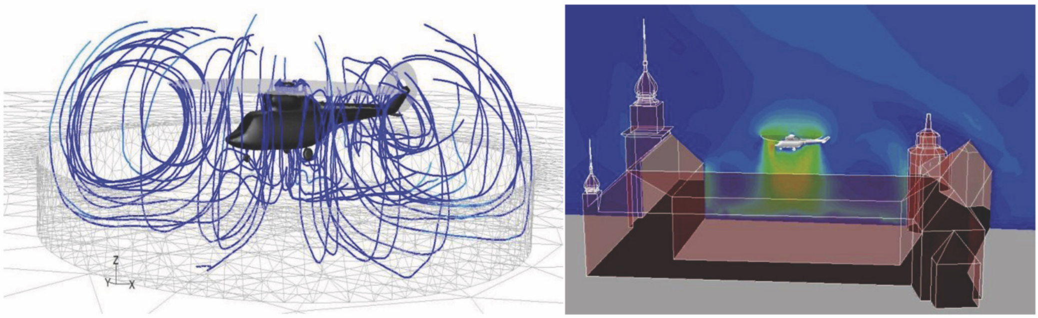

Fig. 16. Right: Helicopter hovers over the well shaped ground (Łusiak et al. 2009) and above the Kings Castle in Warsaw with similar structures visible (Dziubiński et al. 2008)

On the pressure distribution comparison an overpressure area is dissolved in case FAN_WELL_1, as opposed to the other case, FAN_WELL_2, where overpressure area is restricted to small area, which means, that the flow is strongly retarded and can be turned back over the rotor, creating a vortex on the part of rotor, as mentioned above. The same cases comparison of near wall flow velocity shows, that in the first case the stream is closed up in two nearest corridors, and in other case flows smoothly taking all given area. The proof of this analysis is a pathlines visualization, when the looping flow through the rotor is clearly visible in FAN_WELL_1 case. This kind of flow is not observed in the second case.

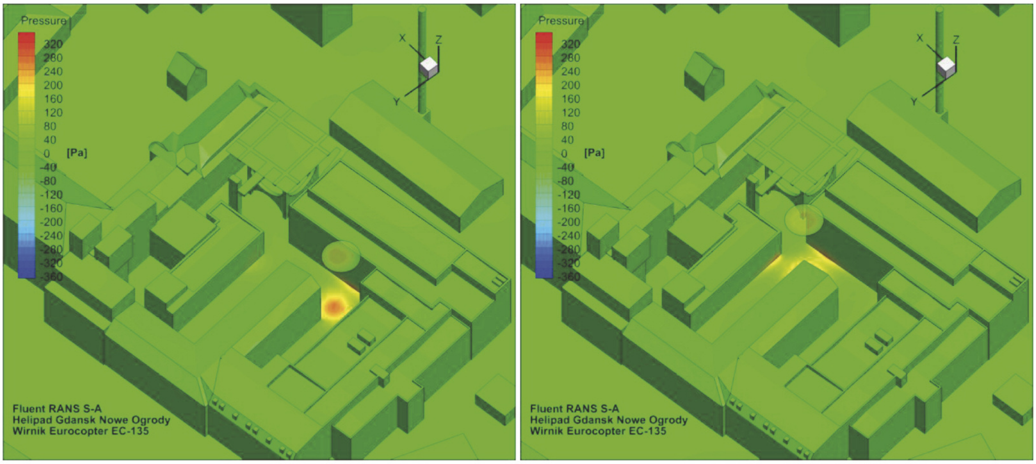

Fig. 17. Static pressure on walls for FAN_WELL_1 i FAN_WELL_2 cases (Dziubiński, 2015)

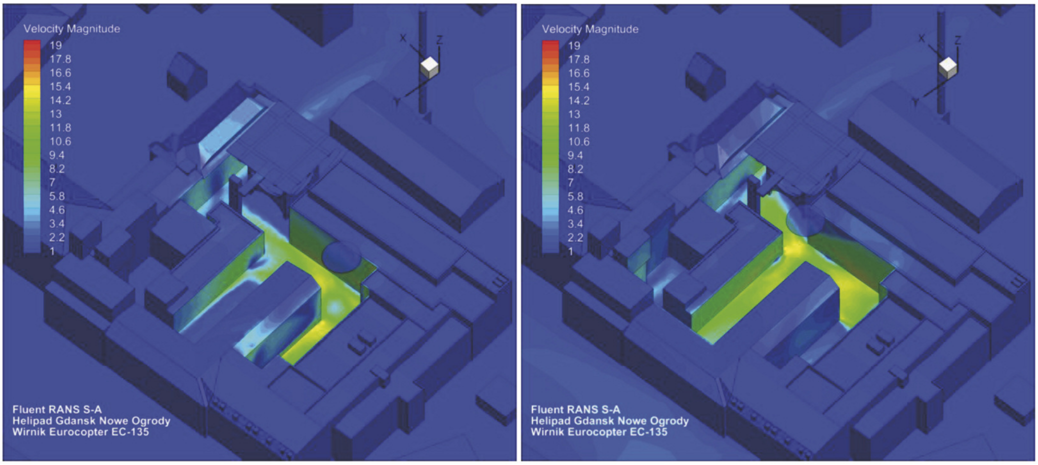

Fig. 18. Near wall velocity for FAN_WELL_1 i FAN_WELL_2 cases (Dziubiński, 2015)

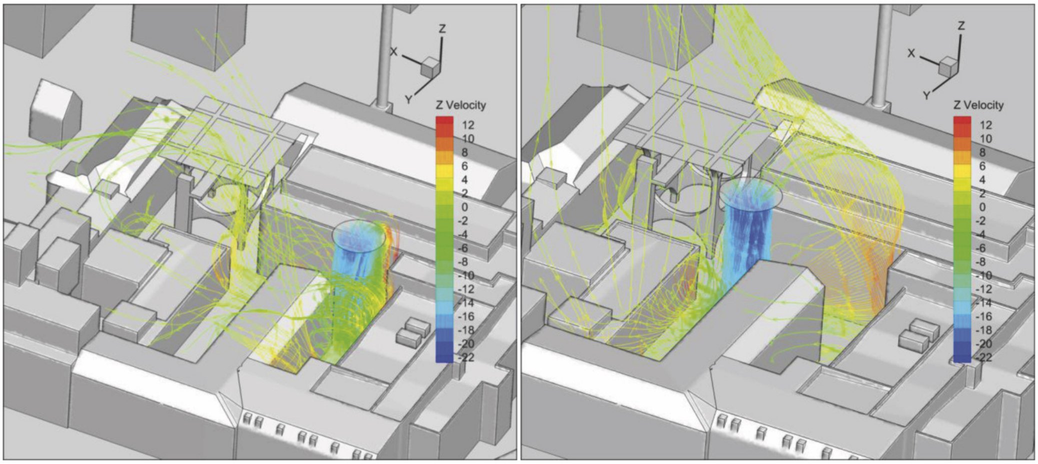

Fig. 19. Pathlines of flow for FAN_WELL_1 i FAN_WELL_2 cases (Dziubiński, 2015)

4. DISCUSSION

During the work a set of test cases has been calculated in order to obtain following information:

- Pressure and velocity distribution, on and near the surface of the building geometry, respectively, caused by the rotor wake during hover over the edge of helipad. An information on flowfoield, influencing approach safety, ground crew safety and expected loads on the buildings, have been shown.
- Difference on flow, especially in terms of pressure load for two sizes of a medevac helicopter.
- The areas where rotor wake could interact with surrounding geometry creating dangerous flow condition have been shown. The helicopters should fly above these areas with significant forward velocity.

The flow on this cases has been calculated, and the results have been compared in terms of pressure distribution near wall velocity, and pathlines. The pressure distribution on the walls is an element taken into account in the design of a helipad. The near wall velocity is useful to provide information about necessity of walkways shielding against the sudden gusts of air. The results proved the following:

- The rain ducts have to be reinforced in close proximity of the helipad, especially in areas exposed to wake.
- Similar analysis could be necessary to assure the ventilation system will sustain the overpressure of 280 Pa, which, appearing on the roof, can pull back the vent stream, disrupt the ventilation and in worst case – introduce the jet fuel fumes back into the hospital vent system.
- For both sizes of a helicopter maximum pressure on the helipad surface has similar value, the difference is only the area. The expected errosion effect on the surface should be bigger on the EC-135 helicopter, since for its rotor the induced velocities are higher.
- The safety condition stating maximum velocity of 17.5 m/s on helipad near the helicopter is barely fulfilled for EC-135, and not fulfilled for Anakonda. It is suggested that the walkways should be covered against it.
- Conserning, that an average person should sustain a wind of 35 m/s, and a helipad has a safety net around it, the above mentioned condition is valid rather for equipment than the ground crew.
- Analysis of hover safety showed, that there is one unsafe area, for which the danger of crash into one of the walls is real. It is recommended that over area named FAN_WELL_1 should be done with sufficient velocity and vertical separation.

The method used on this research proved to be sufficient for wake analysis in number of works. Only RANS method could be used to analyze the rotational flow including aerodynamic interference for such complex geometry. Since the real scale model is calculated, this method should also predict better than wind tunnel model test, a distribution of velocity and pressure distribution caused by the rotor wake.

LITERATURE

[1] ANSYS, 2014, "ANSYS Fluent 14.5 user's guide".

[2] Dziubiński, A., Stalewski, W., and Żółtak, J., 2008, „Przykłady zastosowania pakietu Fluent™ w analizach bezpieczeństwa lotu śmigłowców," Prace Instytutu Lotnictwa, **194-195**, s. 146-157.

[3] Eurocopter, 2014, "Flight Manual of Eurocopter EC-135 P2+," after update.

[4] Łusiak, T., Dziubiński, A., and Szumański, K., 2009, "Interference between helicopter and its surroundings, experimental and numerical analysis," TASK QURTAERLY, **13**(4), pp. 379-392.

[5] Helitech, 2014, "Geodetic and architectural information provided by company".

[6] Sobczak, K., 2008, „Modelowanie wybranych przypadków lotu śmigłowca z wykorzystaniem oprogramowania FLUENT," Prace Instytutu Lotnictwa, **194-195**, s. 158-165.

[7] Świderski, K., 2008, „Modelowanie numeryczne opływu budynków. Wpływ zjawiska konwekcji na pole przepływu," Prace Instytutu Lotnictwa, **194-195**, s. 166-170.

[8] Ruith, M. R., 2005, "Unstructured, Multiplex Rotor Source Model With Thrust And Moment Trimming – Fluent's VBM Model," Fluent VVV COV TN293.

[9] Wikipedia, 2014, „PZL W-3 Sokół," https://pl.wikipedia.org/wiki/PZL_W-3_Sokół.

[10] Dziubiński, A., Grzegorczyk, K., and Żółtak, J., 2011, "CFD Analysis of External Armour Influence on a Helicopter Aerodynamic Characteristics," Transactions of the Institute of Aviation, **218**, pp. 20-27.

[11] Author's unpublished work.

[12] Zhai, Z., Zhang, Z., Zhang, W., and Chen, Q., 2007, “Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part-1: summary of prevent turbulence models,” HVAC&R Research, **13**(6).

[13] Wiśniowski, W., „Instytut Lotnictwa-niestandardowa droga do sukcesu,” Prace Instytutu Lotnictwa, **214**, s. 33-39.

ANALIZA NUMERYCZNA WPŁYWU STRUMIENIA PODWIRNIKOWEGO NA BEZPIECZEŃSTWO UŻYTKOWANIA LĄDOWISK WYNIESIONYCH

Streszczenie

W niniejszej pracy konstrukcja aktualnie projektowanego lądowiska oraz dane dwu przykładowych śmigłowców zostały użyte do zilustrowania metodologii używania obliczeniowej mechaniki płynów do określania bezpieczeństwa użytkowania takich lądowisk. Analiza jest ograniczona do przypadków zawisu, gdzie wpływ strumienia podwirnikowego jest dominujący. Tematem rozważań jest bezpieczeństwo załogi śmigłowca i personelu naziemnego, więc obliczono prędkość przepływu nad płytą lądowiska, rozkłady ciśnień, ale również sprawdzono przepływ indukowany przez strumień podwirnikowy nad obszarami o kształcie studni. Pola przepływu otrzymano przy użyciu oprogramowania analizującego układ równań Naviera-Stokesa (RANS), metodą objętości skończonych, przy użyciu modelu turbulencji Spalart-Allmaras, dlatego że strumień podwirnikowy jest turbulentny a przepływ następuje wokół obiektów o nieaerodynamicznych kształtach. Metoda jest sprawdzona dla przypadków przepływów wirowych z modelowaniem wpływu wytrymowanego wirnika, w symulacjach zawisu w pobliżu budynków. Rezultaty poniższych prac zostały użyte przy projektowaniu wyżej wspomnianego lądowiska.

Słowa kluczowe: Aerodynamika Numeryczna, bezpieczeństwo lotu, lądowisko wyniesione, strumień zaśmigłowy.

Author wishes to thank Mr. Adam Kozłowski (Helitech company) for all help during this research and providing all necessary information. On the same basis author wishes to thank Prof. Kazimierz Szumański (IL) for help in understanding the results and Mr. Leszek Sawicki (State Medical Transport Aviation) for comprehensive information on the helicopter and sharing his practical knowledge. Author also wishes to thank ANSYS Company for the academic associate licenses for their Fluent™ software, and their provider, SymKom company for all their help during calculations.