v. 2, n. 1, 2022

Journal of
Engineering
Research

FATIGUE ANALYSIS OF A REINFORCEMENT CONCRETE TOWER OF AN OFFSHORE WIND TURBINE

Aluizio de Amorim Pacheco
AAJL Engenharia EIRELI
Rio de Janeiro - RJ
http://lattes.cnpq.br/6950737062793646

Renata Gonçalves Faisca
Universidade Federal Fluminense
Niterói - RJ
http://lattes.cnpq.br/3477740011608862

Leonardo Sant`Anna do Nascimento
Bureau Veritas
Rio de Janeiro - RJ
http://lattes.cnpq.br/6641379469332537

Cesar Augusto Salhua Moreno
Universidade Federal de Pernambuco
Recife - PE
http://lattes.cnpq.br/1547813737975557

José Alberto Guimarães Carvalho Viuge
Bureau Veritas
Rio de Janeiro - RJ
http://lattes.cnpq.br/4922635867076076

Jorge Luis Martinez Fernandez
Universidade Federal do Rio de Janeiro
Rio de Janeiro - RJ
http://lattes.cnpq.br/9166558223959202

Abstract: The current research is mainly focused on offshore wind turbines with gravity base foundation, studying the fatigue phenomenon of the structure tower built in reinforcement concrete, which is generally subjected to environmental loads of wave, sea current and wind. The fatigue phenomenon occurs due to the exposure of the structure to cyclic loads, thus, the reinforcement concrete structure, when subjected to environmental loads, can be damaged causing the structure to collapse under stress levels below the maximum project loads. Then, for the structure to withstand the loads imposed during the operational life, it is necessary to carry out fatigue analyzes. The main objective of the research has been to perform a case study, combining environmental loads being applied to a structure of a reinforcement concrete tower of a wind turbine. Currently, there is a search for renewable energy sources that are more attractive in relation to the cost benefit, in this way, the wind has been regarded as one of these sources. Thus, wind turbines have been installed, then it becomes necessary to develop technologies, mathematical models, capable of analyzing the behavior of the structure to allow safe operational activities. For the case study analyzed, first, a global analysis of the system has been performed, using OrcaFlex software, imposing the environmental loads. Subsequently, a local structural analysis has been performed, using ABAQUS Software. In this way, it has been possible to obtain the stresses, which have been used to determine the fatigue life of the tower. With the results obtained it has been possible to conclude that the projected tower can be installed in order to guarantee safe operations at the region of Salvador, Brazil, especially after evaluating the recommended implementations on the structural reinforcement at the base of the wind turbine tower.

Keywords: Wind turbine, fatigue analysis, dynamic offshore analysis, concrete tower.

INTRODUCTION

Nowadays there is a search for renewable energy sources, in this way, the structural behaviour of wind turbines subjected to environmental loads has been studied (GUPTA, BASU, 2020; MAIOLINO, 2014; AMÊNDOLA, 2007).

The offshore wind turbine with **gravity base foundation** was regarded on this work. The tower of the wind turbine was designed regarding two parts. One made of steel and the other made of reinforcement concrete, different from MAIOLINO (2014) work. Then, the main objective of the research was to develop a case study based on the fatigue analysis of the tower (part of reinforcement concrete), regarding the installation in offshore location, at the state of Bahia, Salvador, Brazil.

To analyze the behavior of the structure, after applying static and dynamic loads from the action of waves, current, wind and self-weight, the finite element method was employed, which is capable of estimating the combined loads acting on the structure, which allowed dimensioning the tower by consulting the design standards applied to reinforcement concrete structures (ABNT NBR 6118, 2014; ABNT NBR 6123, 1988; ABNT NBR 15575-1, 2013). Initially, a study was conducted regarding the ANATEMP (2008) software, which is a finite element computational program. In this numerical model, elements of spatial frames were considered, including nonlinear geometric analysis, cross section of a generic structure, and material of concrete and steel reinforcement, in order to verify the combined loads acting on the structure. Thus, with the reactions of the global analyzes (Forces and bending moments) performed, it was possible to design the structure. In addition, another software, OrcaFlex (ORCINA, 1986), was employed to perform global analyzes similar to those performed in ANATEMP (2008). Subsequently, a study was conducted in the ABAQUS (2018) software, through local static analyses, considering physical and geometric nonlinear analysis, thus generating the stresses. Through the stresses obtained, it was possible to compare them with the allowable stresses of reinforcement concrete, thus verifying the design limits, according to ABNT NBR 6118 (2014). Furthermore, in this model, the reinforcement concrete structure was considered with distinct elements for concrete and reinforcement steel.

For the fatigue analysis, in the case study proposed in the current work, local dynamic analyzes were performed in the ABAQUS (2018) software, and from the stresses obtained, a filter was applied to count the cycles of periodic stresses, by the Rainflow method. Finally, it was possible to determine the fatigue life, by assembling the stress range versus number of cycles graphs, the S-N curves. At this stage of the work, a computer program was developed using Visual Basic .Net software. The program was named as ANAFAD (2021). In this program, the Rainflow method was implemented, with the calculation of fatigue life, considering three different methodologies: One based on the DNVGL-RP-C203 (2016) standard, for the steel reinforcement, and two on the DNVGL-ST-C502 (2018) standard, for steel reinforcement and for concrete. For this work, only the two last methodologies were regarded.

METHODOLOGY

The global and local numerical models of the wind turbine developed on this work were based on the Finite Element Method.

To represent the reinforcement concrete of the numerical local model based on the Finite Element Method, and developed in the software ABAQUS (2018), a constitutive model based on the Theory of Plasticity and the Mechanics of Continuous Damage was regarded (COSTA, 2018; ALFARAH, LÓPEZ, OLLER, 2017). Furthermore, concrete stress analysis was evaluated based on the maximum principal stress, and reinforcement steel stress was evaluated based on the von Mises stress.

Moreover, the stress-strain diagram referring to the behavior of the concrete under tension was included in the numerical model. In this way, the numerical model regarded, simultaneously, in the numerical analysis, the two behaviors of the reinforcement concrete (concrete compression and tension), thus, making the model more accurate.

DATA FOR THE NUMERICAL MODELS

All component materials regarded in the present work are shown in Tables 1 and 2.

Table 3 shows the mass and length of the Blades, Rotor and Nacele.

FINITE ELEMENT MODEL DEVELOPED IN THE ANATEMP SOFTWARE (GLOBAL ANALYSIS)

Initially, it was developed a numerical model in ANATEMP (2008) software (Figures 1 and 2).

The parts shown in Figure 2 are described in Table 4.

MODEL CONSIDERATIONS AND BOUNDARY CONDITIONS

- The global reference axis regarded in the model is shown in Figure 1.
- The Structural damping was found to be too low.
- The entire structure, blades, rotor, Nacele and tower was considered as a rigid body. In this way, the values of the modulus of elasticity do not correspond to the real values.

Material	Class	Density (kg/m³)	Young Modulus (GPa)	Tensile Yield Strength (MPa)
Tower (Steel)	ASTM A325	7,850	210	325[a]
Reinforcement Concrete (Steel)	CA-50	7,850	210	500

Note[a]: MAIOLINO (2014).

Table 1: Material data: steel.

Material	Class	Density (kg/m³)	Young Modulus (GPa)	Concrete Strength (MPa)
Concrete	C120[a]	2,548	50.88	120

Note[a]: Adapted from Table 8.1 - ABNT NBR 6118 (2014).

Table 2: Material data: concrete.

Component	Mass (kg)	Length (m)
Pá	14,130	49
Rotor	61,977.57	5
Nacele	81,957.19	20

Table 3: Material data.

Source: Adapted from MAIOLINO (2014).

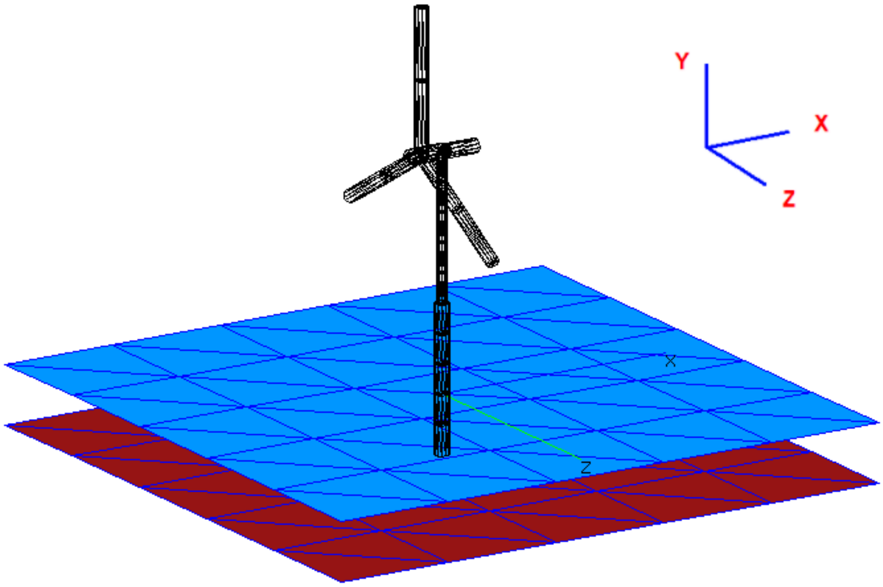

Figure 1: 3D Model in anatemp (2008) Software

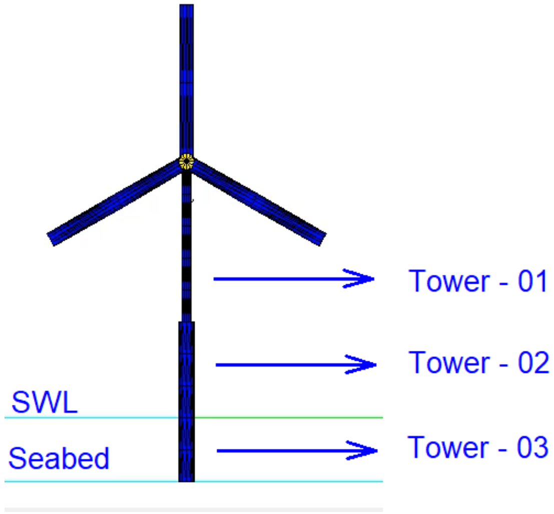

Figure 2: Wind turbine tower parts (ANATEMP, 2008).

Part	Material	Length (m)	OD (m)	ID (m)
Tower - 01	Steel	50	3	2.936
Tower - 02	Reinforcement Concrete	30	5	4
Tower - 03	Reinforcement Concrete	20	5	4

Table 4: Parts of the wind turbine tower.

Source: Adapted from MAIOLINO (2014).

- The structure was considered encastre in the base of the tower.
- Loads regarded in the study: permanent load (own weight), and variable loads: buoyancy, wave, current and wind.
- The loads due to wave, current and wind were considered to act in the same direction, with an azimuth of 0 degrees (North in the X direction).
- The structure of the blades was not considered to be in movement, therefore, the effect of the centrifugal force was not regarded.
- The NACELE swivel mechanism was not considered.
- Marine growth was not considered anywhere in the structure and could be included in future work.
- Water depth of **20 m**.

NUMERICAL ANALYSIS PERFORMED IN THE ANATEMP SOFTWARE (GLOBAL ANALYSIS)

The main information about the numerical analysis performed are presented in Table 5.

ENVIRONMENTAL WAVE LOADING

Wave loading was regarded as a regular wave (Table 6), based on Airy's Linear Theory. The water depth was considered to be **20 m**.

WIND LOADING

The wind action was considered as a static load acting on the turbine blades and tower, with a wind speed of **12 m/s** (MAIOLINO, 2014).

CURRENT LOADING

The action of the current was considered as a rectangular loading profile, evenly distributed from the seabed to the surface of calm waters, with a speed of **0.70 m/s** (MAIOLINO, 2014).

TIME DOMAIN DYNAMIC ANALYSIS RESULTS

Table 7 presents the reaction forces and bending moments obtained as a result of the dynamic analysis performed in the time domain, on the encastre base of the wind tower.

PRE-DIMENSIONING

After the dynamic analysis, the pre-dimensioning of the tower (Reinforcement concrete), **Tower - 02** and **Tower - 03**, was carried out (Figures 3 and 4). The calculation procedures were based on the ABNT NBR 6118 (2014) standard. Thus, after some simplified calculation attempts, just to estimate an initial value of the steel area, it was concluded that the tower should have a thickness of **1 m**. Initially, a thickness of **0.5 m** had been estimated, as can be deduced from the information in Table 4.

FINITE ELEMENT MODEL DEVELOPED IN THE ORCAFLEX SOFTWARE (GLOBAL ANALYSIS)

After the development of the previous Section, which was the initial phase of the work, it was developed analysis in the OrcaFlex software (ORCINA, 1986). Then, it was possible to execute analysis regarding a total time (**10,800 s**) bigger than the analysis executed in ANATEMP (2008) (**150 s**). The ANATEMP software executes offshore analysis, however, it takes much more time to complete the analysis. The ANATEMP (2008) is not a commercial software, like OrcaFlex (ORCINA, 1986).

Figure 5 shows the numerical model developed in OrcaFlex software.

The parts shown on Figure 5 are described in Table 8. It differs from Table 4 with respect to the inner diameter of the **Tower - 02**, which was updated as per indicated on Section 5.

Journal of Engineering Research ISSN 2764-1317 DOI 10.22533/at.ed.317212210012

Type of Analysis	Time Step (s)	Total Time (s)
Dynamic	0.1	150

Table 5: Numerical analysis data.

Type of Wave	Wave Height (m)	Period (s)
Regular	4.4	7.5

Table 6: Environmental wave loading (MAIOLINO, 2014).

Time (s)	FX (kN)	FY (kN)	MZ (kN.m)
148	-932.75	9,708.1	44,573.00

Table 7: Forces and bending moment of reaction.

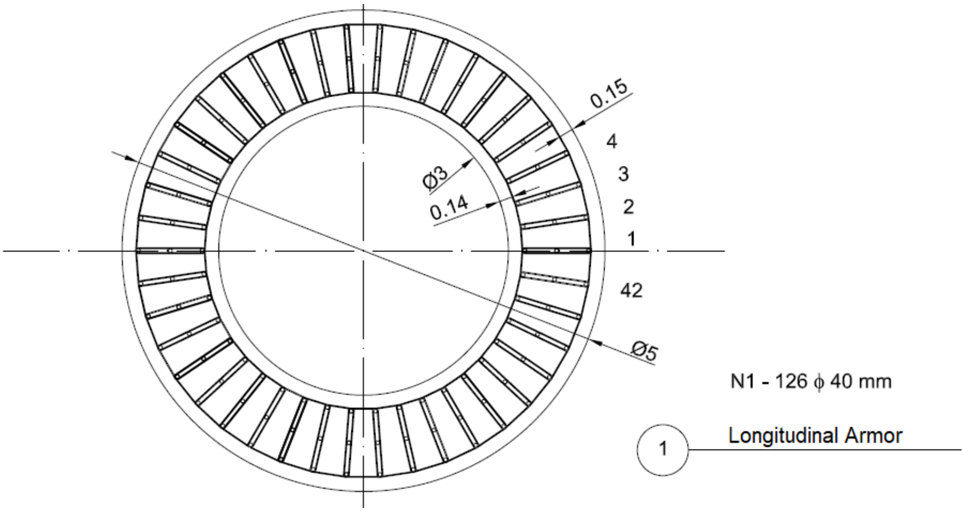

Figure 3: Longitudinal reinforcement.

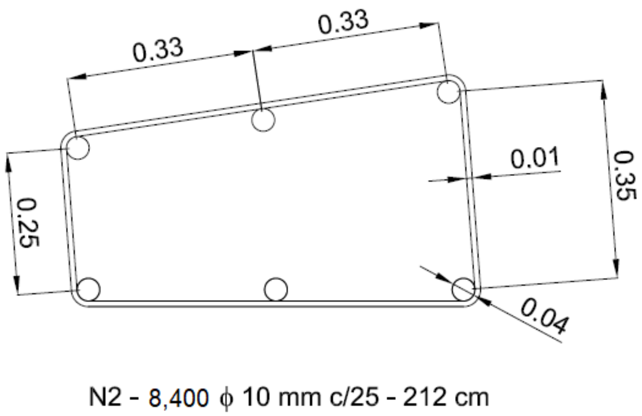

Figure 4: Transversal reinforcemente.

Journal of Engineering Research ISSN 2764-1317 DOI 10.22533/at.ed.317212210012

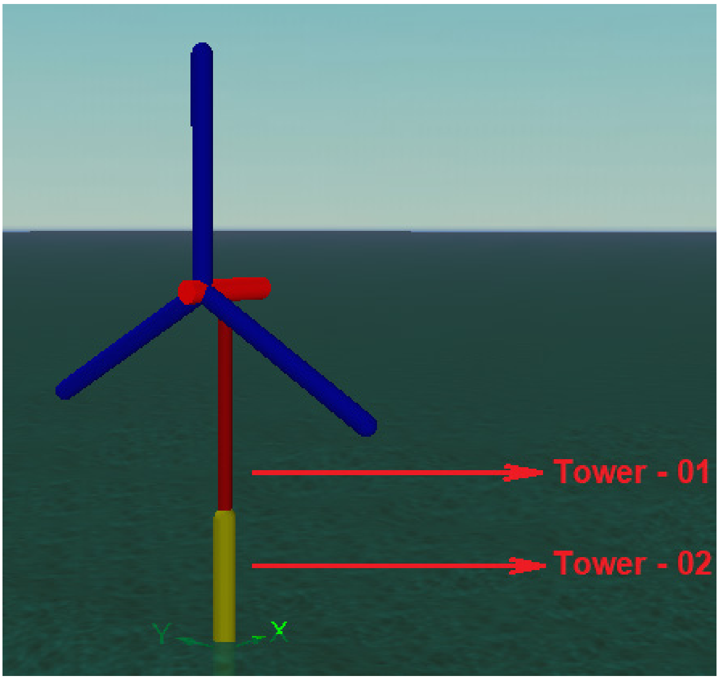

Figure 5: 3D model in orcaflex software (ORCINA, 1986).

Part	Material	Length (m)	OD (m)	ID (m)
Tower - 01	Steel	50	3	2.936
Tower - 02	Reinforcement Concrete	50	5	3

Table 8: Parts of wind turbine tower.

MODEL CONSIDERATIONS AND BOUNDARY CONDITIONS

The same comments described on Section 3.1 are applied for the current Section.

NUMERICAL ANALYSIS PERFORMED ON ORCAFLEX (GLOBAL ANALYSIS)

The main information about the numerical analysis performed are presented in Table 9.

LOADING CASE MATRIX

The loading case matrix is shown in Table 10. Environmental data were extracted and adapted from the references MAIOLINO (2014), CAMPOS (2009), PIRES (2017) and SILVA (2013). The azimuth was regarded as **0 degree**, and the inner diameters of **Cases 04** and **05** were considered as **3 m**. For the other Cases, **4 m**. However, the results were similar.

TIME DOMAIN DYNAMIC ANALYSIS RESULTS

Table 11 presents the reaction forces and bending moments obtained as a result of the dynamic analysis performed in the time domain, on the encastre base of the wind tower.

Comparing the answers between **Case 01**, Tables 11 and 7. It is clear that there is a difference, especially for the bending moment. As this discussion is not part of the present work, it may be discussed in future works.

FINITE ELEMENT MODEL DEVELOPED IN THE ABAQUS SOFTWARE (LOCAL ANALYSIS)

To verify the strength of the structure designed in Section 5, a numerical model was developed in the ABAQUS (2018) software, with the objective of obtaining the stresses for the structure in concrete and for the structure of the longitudinal and transverse reinforcements, and then compares to the allowable stresses of the concrete and the armors. Figure 6 shows the schematic of the numerical models developed in ABAQUS (2018) software.

Due to simplifications in the numerical model, the **Model (B)** was developed, which represents a part of the section of the tower in reinforcement concrete (Slice of **Model A**), with a height of **5 m**. The blue part represents the concrete, and the red part, the longitudinal and transversal reinforcement. The local model is more realistic than the global (OrcaFlex Model), once the structure is represented with the concrete and the steel reinforcement separately, in different finite elements, but linked in a proper way, thus, representing the stiffness of the global structure more accurately.

The finite element discretization of the reinforcement concrete structure was defined as shown in Table 12.

MODEL CONSIDERATIONS AND BOUNDARY CONDITIONS

- The global reference axis considered in the model is shown in Figure 6.
- The structure was regarded encastre in the base of the tower (Figures 7).
- Loads were distributed in two ways. **Case 01**: The force was applied to the upper surface of the model. And **Case 02**: force was applied to the lateral surface of the structure (Figure 7).

For the model **Model B** - **Case 01**, from Load **Case 03**, the ultimate limit state was not satisfied according to ABNT NBR 6118 (2014). In the ideal structural model, the loads are distributed along the structure, then, the alternative model (Figure 7) was developed.

NUMERICAL ANALYSIS PERFORMED IN ABAQUS (LOCAL ANALYSIS)

The main information about the numerical analysis performed are presented in Table 13.

Type of Analysis	Time Step (s)	Total Time (s)
Dynamic	0.1	10,800

Table 9: Numerical analysis data.

Case	Type of Wave	H (m)	T (s)	Current (m/s)	Wind (m/s)
01	Regular - Airy	4.40	7.50	0.70	12.00
02	Irregular - JONSWAP	10.00	8.33	1.00	12.00
03	Irregular - JONSWAP	12.00	8.33	1.00	20.00
04	Irregular - JONSWAP	12.00	8.33	1.00	20.00
05	Irregular - JONSWAP	10.00	8.33	1.00	30.00

Table 10: Loading case matrix.

Case	FX (kN)	FZ (kN)	MY (kN.m)
01	-993.96	6,812.99	20,101.63
02	-4,598.46	8,479.99	89,190.00
03	-6,540.54	10,091.04	140,850.00
04	-6,556.84	15,535.08	144,220.00
05	-5,150.82	14,932.40	138,840.00

Table 11: Forces and bending moment of reaction.

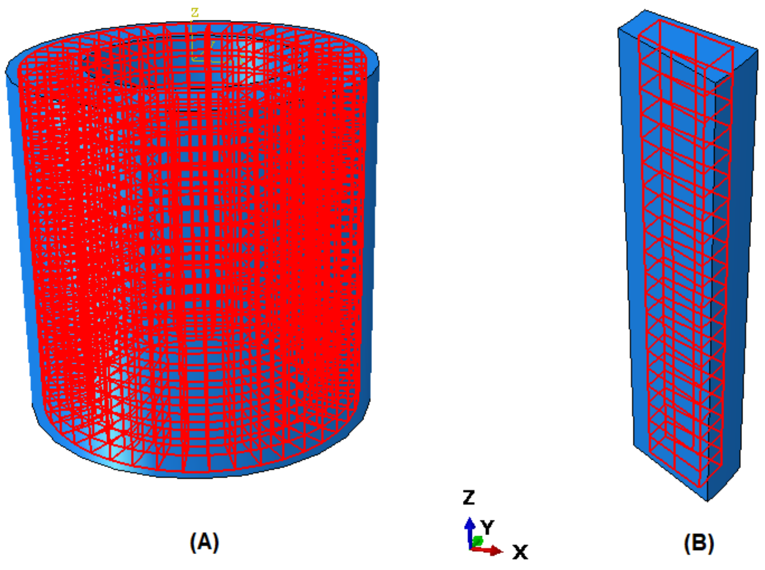

Figure 6: 3D Model in abaqus (2018) software.

Component	Type of Element in ABAQUS	Length (m)
Concrete	C3D8R - 8 linear nodes	0.10
Longitudinal Reinforcement	T3D2 - 2 linear nodes	0.05
Transversal Reinforcement	T3D2 - 2 linear nodes	0.05

Table 12: Finite element mesh.

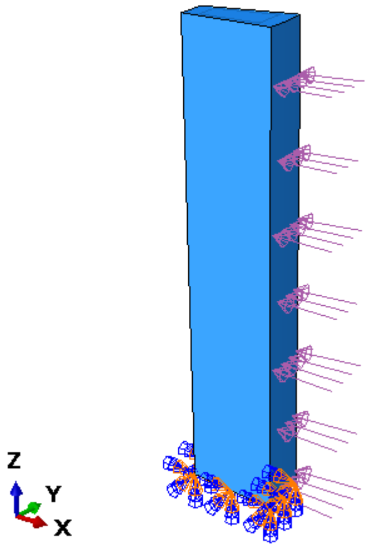

Figure 7: Force applied on the lateral of the structure (ABAQUS, 2018).

Type of Analysis	Time Step (s)	Total Time (s)
Static	0.1 / 0.001	100

Table 13: Numerical analysis data.

LOADING CASE MATRIX

The loading case matrix is shown in Table 14, for the type of load shown in Figure 7. The equivalent forces per unit of area (Second column of Table 14) were extracted from Tables 7 and 11. As a local analysis was carried out, with a representative model of the global system, the forces were obtained proportional to the area of the model. The reference global model was idealized with a lateral load area corresponding to a **90 degree** arc and a height of **50 m** (height of the reinforcement concrete part of the tower).

The load Cases for the **Model (A)** were not presented because the Model was not executed in ABAQUS (2018), once the static analysis was taking a long period of computational time. It was estimated more than **17 days** to complete the analysis in a **Notebook i5**, processor of **2.6 GHz** and **8 Gb** of Ram memory.

ELASTO-PLASTIC MODEL

Figures 8, 9 and 10 (fissuration strain) show the stress versus strain diagrams considered in the numerical analyses, for the two materials, concrete and steel (Longitudinal and Transversal reinforcement).

STATIC ANALYSIS RESULTS: MODEL (B)

Tables 15 (Concrete) and 16 (Steel reinforcement) present the responses regarding loads applied as shown in Figure 7.

The stresses presented in Table 15 are the maximum principal stresses. Moreover, the “**NA**” text refers to the numerical simulations that were not able to converge.

The allowable stresses for concrete on compression and tension, and for steel reinforcement are presented in Table 17.

Verifying the stresses shown in Tables 15 and 16, it can be concluded that they are in accordance (≤) with the allowable stresses presented in Table 17. About the displacements, taking as a limit the value **L/250** (ABNT NBR 6118, 2014, Table 3.13), for the service limit state, it can be concluded that the results are in accordance with the standard, once **5 m/250 = 0.02 m** is greater than the values presented in the Table 15.

For the simulations where the results were classified as “**NA**”, unconverged, probably, a **reinforcement at the base of the structure** will ensure that the structure will resist the applied loads, thus favoring the convergence of the numerical response. The reinforcement at the base of the structure is a normal procedure in the design, and can be regarded in future works.

The stresses for concrete on compression were extracted from the part of the structure shown in Figure 11.

The stresses for concrete on tension were extracted from the part of the structure shown in Figure 12.

The stresses for steel (Longitudinal and transversal reinforcement) were extracted from the part of the structure shown in Figure 13.

FATIGUE ANALYSIS

For the fatigue analysis, a simplification was done in order to reduce the analysis run time, once the aleatory data from de global analysis, equivalent force, is composed by **107,500 points**. Thus, it was decided to run analyzes considering periodic forces with the amplitudes being represented by the maximum values obtained from the temporal series of the aleatory data from the global analysis (Equivalent forces). It is understood that the adopted strategy represents the worst case, because it was regarded the maximum value of equivalent force for the amplitude. Thus, the time step of **0.1 s**, and **1,001 points** were adopted for the dynamic local analysis. The loads were applied as shown in Figure 7.

Case	Equivalent Force (kN/m²)
01 (Table 7)	9.28
02 (Table 11)	45.96
03 (Table 11)	67.90
04 (Table 11)	68.41
05 (Table 11)	53.93

Table 14: Load case matrix.

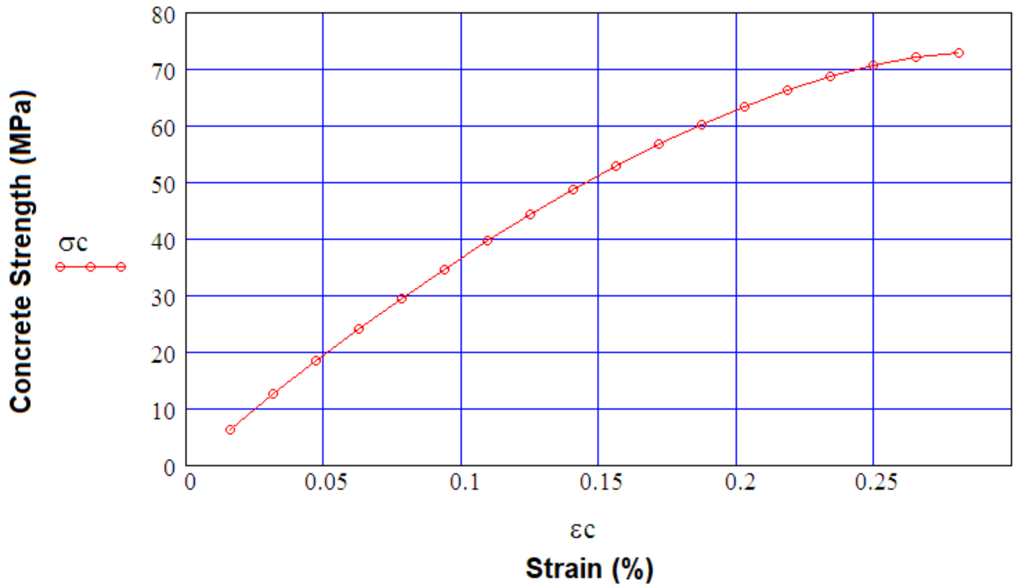

Figure 8: Stress-strain concrete on compression (ABNT NBR 6118, 2014).

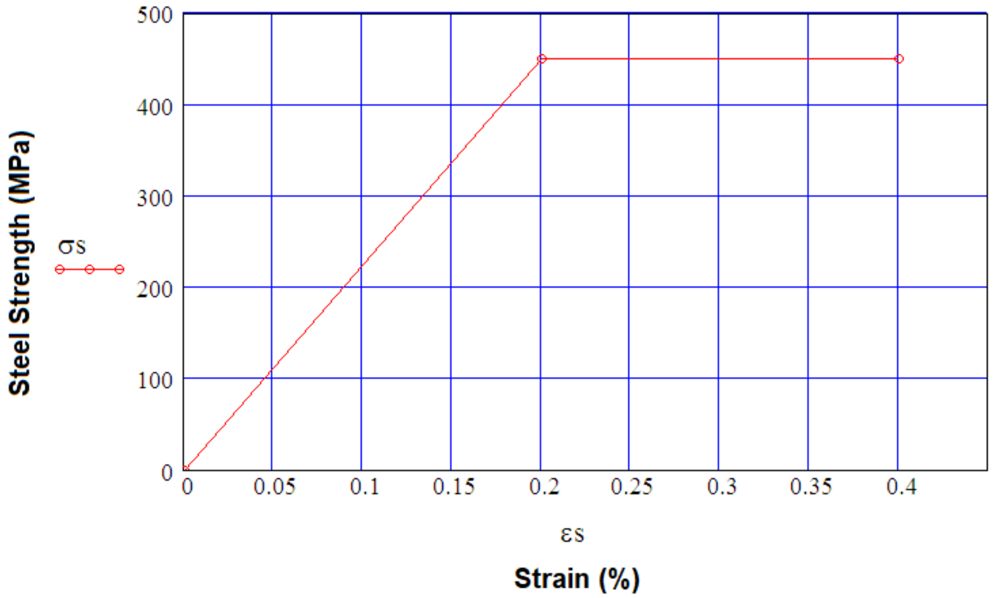

Figure 9: Stress-strain steel on tension (ABNT NBR 6118, 2014).

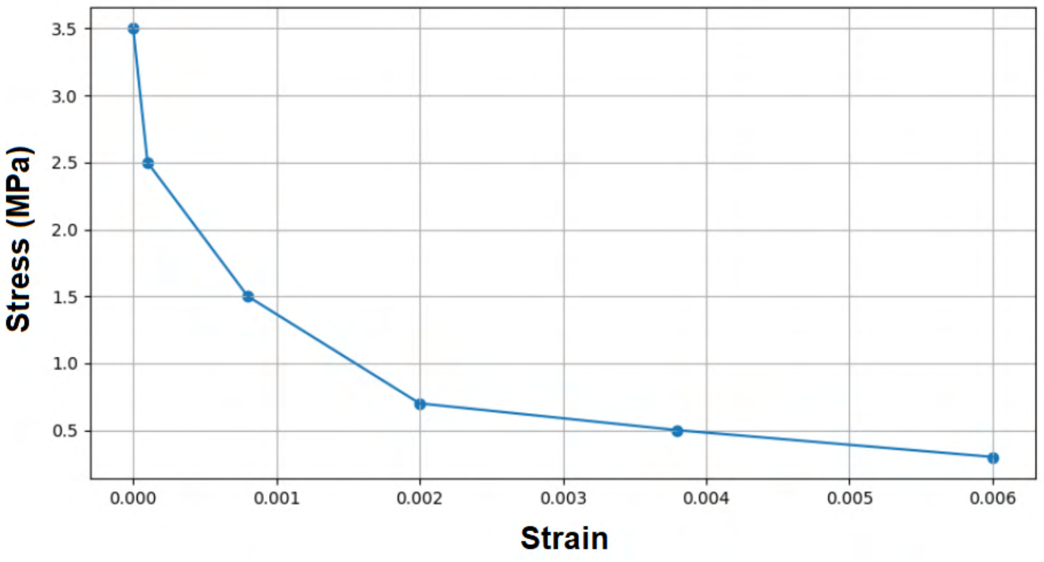

Figure 10: Stress-fissuration strain for concrete on tension (COSTA, JÚNIOR, JÚNIOR, 2018; ALFARAH, LÓPEZ, OLLER, 2017).

Case	Stress on Compression (MPa)	Stress on Tension (MPa)	Displac. Max. (m)	Run Time (s)
01	0.072	0.74	0.00022	33
02	0.39	3.47	0.0011	32
03	**NA**	**NA**	**NA**	**NA**
04	**NA**	**NA**	**NA**	**NA**
05	0.44	3.54	0.0013	44

Table 15: Results for concrete.

Case	Equivalent Force (kN/m²)	von Mises Stress (MPa)
01	9.28	2.3
02	45.96	11.74
03	**67.90**	**NA**
04	**68.41**	**NA**
05	53.93	19.34

Table 16: Results for steel reinforcement.

Criterion	Allowable Stress (MPa)	Standard
Stress on Comp. in Concrete	72.86	ABNT NBR 6118, 2014 - Table 12.1
Stress on Tension in Concrete	3.94	ABNT NBR 6118, 2014 - Table 12.1
Stress in Steel	434.78	ABNT NBR 6118, 2014 - Item 8.2.5

Table 17: Allowable stress.

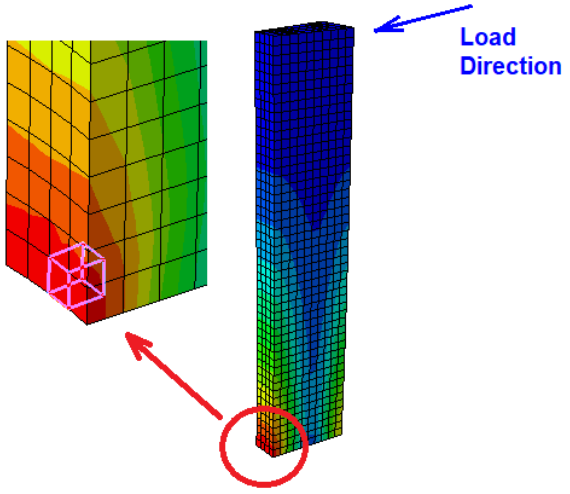

Figure 11: Concrete on compression.

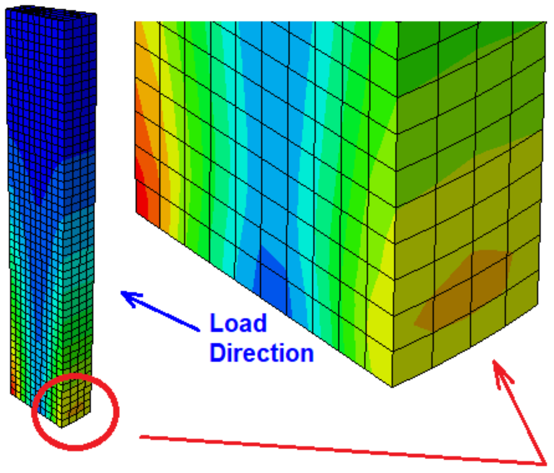

Figure 12: Concrete on tension.

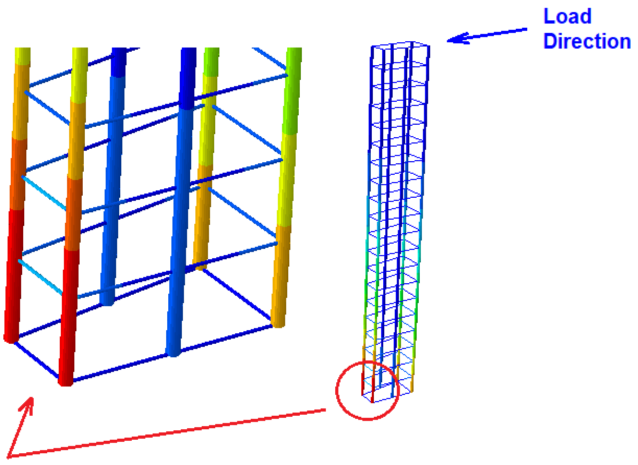

Figure 13: Stresses (reinforcement).

The loading case matrix is shown in Table 18.

FATIGUE ANALYSIS RESULTS: MODEL (B)

The results for Fatigue Analysis are shown in Tables 19 up to 22.

In Table 19, the fatigue life with respect to concrete was presented, indicating a fatigue life of **2.09E+14 years** (**Case 1**), that is, a very long life, regarding as criterion a service life equal to **50 years** (ABNT NBR 15575, 2013), and a safety factor equal to **10** (**500 years**). Probably the high fatigue life value was due to the fact that the maximum stress range was low. The same comment is valid for **Cases 2** and **5**.

In Table 20, the fatigue life was presented with respect to steel reinforcement, indicating a infinite fatigue life (**Case 01**), that is, a very high life, regarding a fatigue life criterion for steel reinforcement equal to **50 years** (ABNT NBR 15575, 2013), and a safety factor equal to **10** (**500 years**). Probably the high fatigue life value occurred due to the fact that the stress range is low for the steel bars adopted. The same comment is valid for **Cases 2** and **5**.

In Table 21, a verification was made considering the formulation presented in ABNT NBR 6118 (2014), for concrete subjected to tensile conditions. For the evaluation of concrete subjected to compression conditions, it is suggested that this analysis be carried out in future works. Thus, for **Case 1**, it can be concluded that the allowable stress limit (**0.84 MPa**) was exceeded by the maximum stress obtained in the dynamic analysis (**0.85 MPa**), thus not passing on the fatigue verification. Moreover, for **Cases 2** and **5**, it is verified that the maximum stresses also exceeded the limit allowable in the standard, thus, not being approved in the fatigue verification. It is important to highlight that the formulation adopted (ABNT NBR 6118, 2014 - Item 8.2.5) is valid for concrete of the class **C55** up to **C90**, and the concrete regarded on the project is of the class **C120**, thus, the fatigue limit (**fctd,fad**), probably, was overestimated. Therefore, it is recommended to consider a more adequate formulation to estimate the "**fctm**", which is valid for concrete of class **C120**. This implementation can be done in future works. Furthermore, the stress versus strain diagram for concrete under tension does not correspond to the concrete class defined in the structure design. The diagram considered in the analysis corresponds to class **C25**, while for the project, the class **C120** was defined. Thus, the tensile strength values were underestimated. Probably, higher stresses values occurred and exceeded the limit "**fctd,fad**". Therefore, for future work, it is recommended that the numerical simulations be conducted considering a stress-strain diagram (Concrete under tension) compatible with **C120** concrete.

Moreover, it is necessary to **re-evaluate** the structure considering a **reinforcement at the base of the structure**, making the numerical model closer to reality. Remembering that the highest stress values occurred at the base of the structure, which was the defined location for obtaining the stresses. That reinforcement at the base of the structure probably will lead the structure to have lower stress values.

In Table 22, a verification was made considering the formulation presented in ABNT NBR 6118 (2014), for the steel reinforcement. Thus, for **Case 1**, it can be concluded that the allowable stress limit (**65.00 MPa**) was not exceeded by the von Mises stress obtained from the dynamic analysis (**2.30 MPa**), thus, being approved in the verification for fatigue. The same comment is valid for **Cases 2** and **5**.

Case	Equivalent Force (kN/m²)	Period (s)
01	9.28	7.5
02	45.96	8.33
05	53.93	9.18

Table 18: Load case matrix.

Case	C1	C5	Max. Stress Range (MPa)	Fatigue Life (Years)	Run Time (HMS)	Check
01	8	1	0,93	2.09E+14	00:14:53	**Ok**
02	8	1	3,75	2.07E+13	00:17:18	**Ok**
05	8	1	3,43	2.26E+13	00:20:17	**Ok**

Note 1: Fatigue life for reinforcement concrete equal to 50 years (ABNT NBR 15575, 2013).

Note 2: Safety factor of 10. Limit of 500 years.

Table 19: Concrete (DNVGL-ST-502, 2018).

Case	C3	C4	Max. Stress Range (MPa)	Fatigue Life (Years)	Run Time (HMS)	Check
01	19,6	6	2,30	Infinity	00:14:53	**Ok**
02	19,6	6	16,93	1.01E+6	00:17:18	**Ok**
05	19,6	6	23,06	182,354.05	00:20:17	**Ok**

Note 1: Fatigue life for reinforcement concrete equal to 50 years (ABNT NBR 15575, 2013).

Note 2: Safety factor of 10. Limit of 500 years.

Table 20: Steel reinforcement (DNVGL-ST-502, 2018).

Case	Min Stress (MPa)	Max Stress (MPa)	□f . □ct,max (MPa)	fctd,fad (MPa)	Run Time (HMS)	Check
01	-0,07	0,85	0,85	0,84	00:14:53	**Not Ok**
02	-0,18	3,57	3,57	0,84	00:17:18	**Not Ok**
05	0,00	3,44	3,44	0,84	00:20:17	**Not Ok**

Note 1: Concrete in traction.

Note 2: Coefficient □f = 1 obtained from item 23.5.3 of ABNT NBR 6118 (2014).

Note 3: Parameter fctd,fad = 0.3 . fctd,inf, obtained from item 23.5.4.2 of ABNT NBR 6118 (2014).

Table 21: Concrete (ABNT NBR 6118, 2014).

Case	Min Stress (MPa)	Max Stress (MPa)	□f . □□Ss (MPa)	□fsd,fad (MPa)	Run Time (HMS)	Check
01	0,00	2,30	2,30	65	00:14:53	**Ok**
02	0,00	17,12	17,12	65	00:17:18	**Ok**
05	0,00	23,20	23,20	65	00:20:17	**Ok**

Note 1: Coefficient □f = 1 obtained from item 23.5.3 of ABNT NBR 6118 (2014).

Note 2: Parameter □fsd,fad obtained from Table 23.2 of ABNT NBR 6118 (2014) (Item 23.5.5).

Table 22: Steel reinforcement (ABNT NBR 6118, 2014).

CONCLUSION

Both the results related to the ultimate limit state and service limit state were favorables to the limits of the standards regarded in the present work. For unconverged results (Tables 15 and 16), it is important to mention that, generally, the type of structure regarded, offshore wind turbine with gravity base foundation, includes in the design, a reinforcement at the base of the structure, thus, an implementation on the numerical model regarding the reinforcement, probably, will distribute the loads more adequately, consequently, it will reduce the levels of stresses, and finally it will contribute to converge the numerical solution, specially, in the regions of stress concentration.

For the unfavorables results presented in Table 21, it is important to highlight that the formulation adopted is valid for concrete of the class **C55** up to **C90**, and the concrete defined in the structure design is of the class **C120**, thus, the fatigue limit (**fctd,fad**), probably, was overestimated. Therefore, it is recommended to regard a formulation to estimate the "**fct,m**", which is valid for concretes of the class **C120**. This procedure can be done in future works. The same comment is valid for the formulation of the stress-strain diagram of concrete on tension, which is based on concrete of class **C25**, while the structure design was defined with the concrete class of **C120**.

ACKNOWLEDGEMENTS

The authors would like to thank the Fluminense Federal University, the Federal University of Pernambuco, the companies Bureau Veritas and AAJL Engenharia EIRELI for the technical-scientific contribution of their collaborators in the preparation of the current work.

REFERENCES

ABAQUS, **Finite Element Structural Analysis Software**, 2018.

ABNT NBR 6118 - **Projeto de estruturas de concreto** - Procedimento, Brasil, 2014.

ABNT NBR 6123 - **Forças devidas ao vento em edificações** - Procedimento, Brasil, 1988.

ABNT NBR 15575-1 - **Edificações habitacionais - Desempenho**, Parte 1: Requisitos gerais, 2013.

ALFARAH, B., LÓPEZ-Almansa, F., OLLER, S., **New methodology for calculating damage variables evolution in Plastic Damage Model for RC structures**. Engineering Structures, Elsevier, v. 132, p. 70-86, 2017.

AMÊNDOLA, C. A. M., **Contribution to the Study of Variable Speed and Pitch Wind Turbines with Dual Powered Generator and Diffuse Control System**. Doctoral Thesis, School of Engineering of São Carlos, University of São Paulo, São Carlos, 2007.

ANAFAD, **Fatigue Analysis Software**, 2021.

ANATEMP, **Finite Element Structural Analysis Software**, 2008.

CAMPOS, R. M., **Analysis of wave extremes in Rio de Janeiro associated with extratropical cyclones in the South Atlantic**. Master's Dissertation, Postgraduate Program in Oceanic Engineering, COPPE, Federal University of Rio de Janeiro, Rio de Janeiro, 2009.

COSTA, M. H. A., JÚNIOR, M. S. M., JÚNIOR, E. P., **Analysis of Reinforced Concrete Structures Considering Damage and Plasticity**. Symposium on Computational Mechanics, Federal University of Espírito Santo (UFES), Vitória, 2018.

DNVGL-RP-C203, **Fatigue design of offshore steel structures**. Norway: Det Norske Veritas, 2016.

DNVGL-ST-C502, **Offshore concrete structures**. Norway: Det Norske Veritas, 2018.

GUPTA, B. K., BASU, D., "**Offshore wind turbine monopile foundations: Design perspectives**". Elsevier Science, Ocean Engineering, 107514, 2020.

MAIOLINO, P., **Dynamic analysis of a monopile offshore wind turbine**. Master's Dissertation, Postgraduate Program in Civil Engineering, COPPE, Federal University of Rio de Janeiro, Rio de Janeiro, Brazil, 2014.

ORCINA, **Offshore Finite Element Structural Analysis Software**, 1986.

PIRES, L. B., **Study of currents for the dry and rainy periods of the northern coast of Bahia**. Monograph presented to the oceanography course, Institute of Geosciences, Federal University of Bahia, Salvador, 2017.

SILVA, N. P., **Wind Extremes Over the West of the South Atlantic Ocean: Directional Analysis of Occurrences**. Institute of Astronomy, Geophysics and Atmospheric Sciences, São Paulo, 2013.