Prediction of fatigue damage by random vibration using isogeometric and finite element analysis

Shubiao WANG[1], Renata_Troian[1], Leila Khalij [1]

[1]INSA Rouen Normandy, France

ABSTRACT

At present, the Finite Element Analysis (FEA) method is indispensable in the field of simulation technology, as this kind of numerical analysis method can assist engineers to predict results, which are often difficult to obtain from experimental tests. However, there exist some problems in terms of finite element mesh generation time and geometric representation. In this studying, we adopted a new numerical analysis method, Isogeometric Analysis (IGA) to develop static and dynamic analyses on two models, a notched plate and a wind turbine tower model in Ls Dyna software. From the static convergence analysis result, it is shown that IGA is more time-efficient compared with FEA. In terms of fatigue analysis results, IGA can predict the fatigue life corresponding very well to the fatigue life computed by FEA. It can be concluded that IGA is appropriate for the numerical analysis.

Keywords: Isogeometric analysis; Finite element method; random vibration fatigue analysis

1 Introduction

At present, there are several disadvantages in classical FEA. The most significant one is to spend a long time in mesh generation. For example, it is estimated that about 80% of overall analysis time has been applied to the mesh creation process in automotive, aerospace, and shipbuilding industries [1]. In 2005, T.J.R. Hughes proposed a method, which was named Isogeometric analysis (IGA) to mainly solve the problems derived from the classical FEA.

IGA with NURBS basis function has been applied in various engineering problems, including contact mechanics [2-4], fluid mechanics [5-7], structural optimizations [8-11], shell analysis [12-15], beam analysis [16, 17, 20], damage and fracture mechanics [18, 19], and structural vibration analysis [16, 20, 21], etc. In this paper, we mainly investigate the performance of the NURBS-based IGA in LS-DYNA on a plate and wind turbine tower model. Results were verified by classical FEA and matlab code.

The originality of this paper is that the isogeometric random vibration fatigue analysis is firstly employed on an industrial model. The structure of this article is as follows. In section 2, isogeometric random static and fatigue analysis is applied on a plate model. In section 3, isogeometric random vibration fatigue analysis is developed on a wind turbine tower model, and the results are verified by the FEA and own developed Matlab programming. In section 4, we conclude on the present studies.

2 IGA procedures in Ls Dyna

In this section, I will mainly present the isogeometric static and dynamic analysis processes in Ls Dyna with the use of a plate model analyzed in this studying.

2.1 Geometric model construction

Using the keywords, PLANE and CIRCLE (Fig. 1 (a) and (b)), a plate and two circles with the length, width and radius of 0.08, 0.02 and 0.010876 m can be constructed, as shown in Fig.1 (d). Then, based on the keyword, TRIMMING (Fig. 1 (c)), the final plate model can be obtained, as the model in Fig. 1 (e).

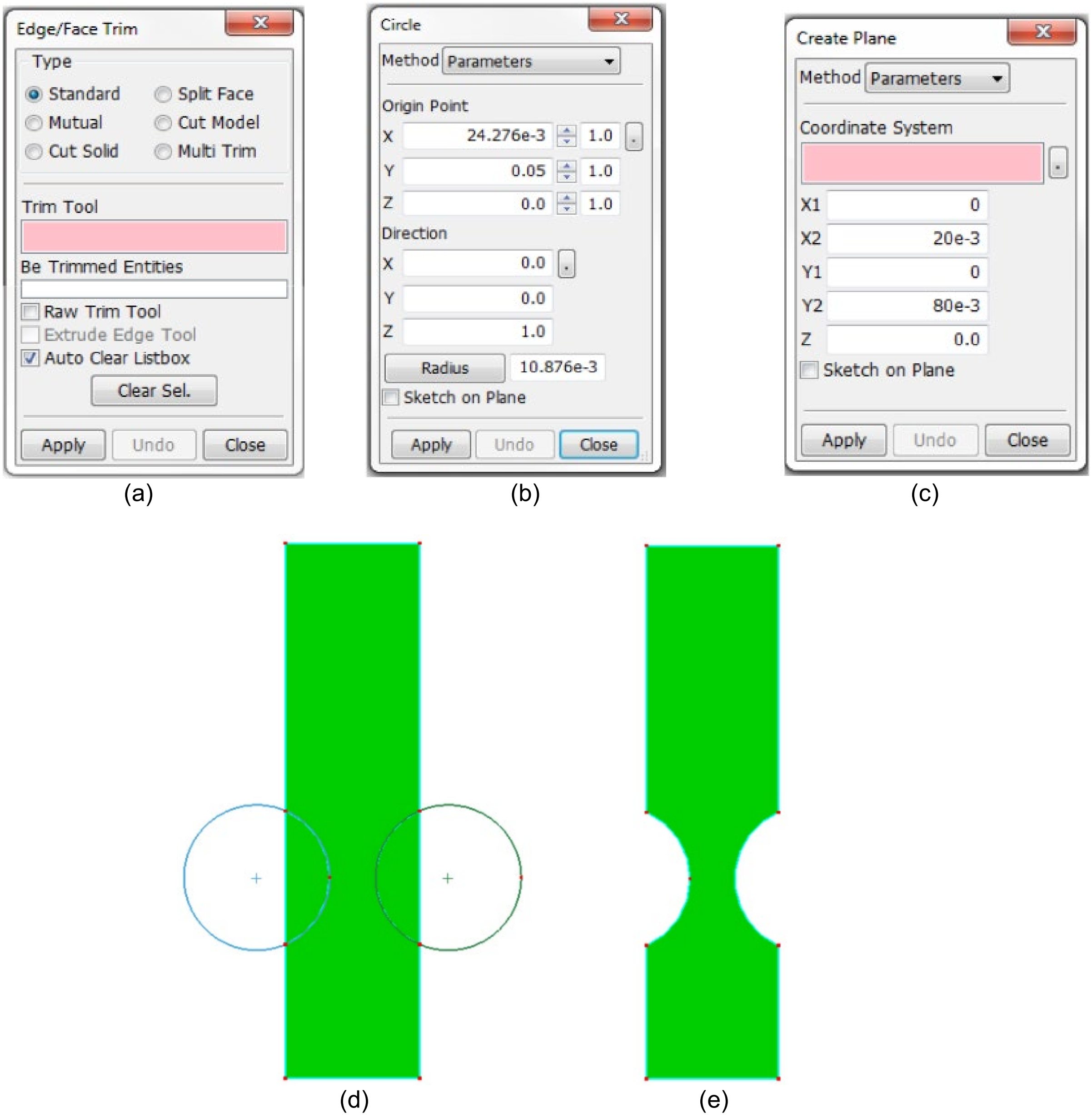


Fig.1: The geometry construction (a) Plate construction; (b) Circle construction; (c) Geometry trimming; (d) Created model; (e) Final mode).

2.2 Mesh model construction

Using the keyword, NURBS EDITOR, in the first step, the rough NURBS model can be created, which consists of 9 control points, and polynomial order of univariate NURBS basis functions in x and y-directions are 2 (Fig. 2 (a)). In the second step, using the keyword, REFINE and SUBDIVIDE Fig. 2 (b), the rough mesh model obtained from the first step is refined, in which the segment is equal to the NURBS element, and in r (x) and s (y)-directions, the segment numbers are respectively set 5 and 20. The final mesh model is shown in Fig. 2 (c), it's comprised of 154 control points in this case, and polynomial order of NURBS basis functions in both directions are respectively 2. The detail information related to the mesh model can be observed from the Fig. 3, for example, it can be seen that the number of control points in r (x) and s (y)-are 7 and 22, corresponding polynomial orders are 2, and the knot vector in r direction is {0, 0, 0, 0.2, 0.4, 0.6, 0.8, 1, 1, 1}.

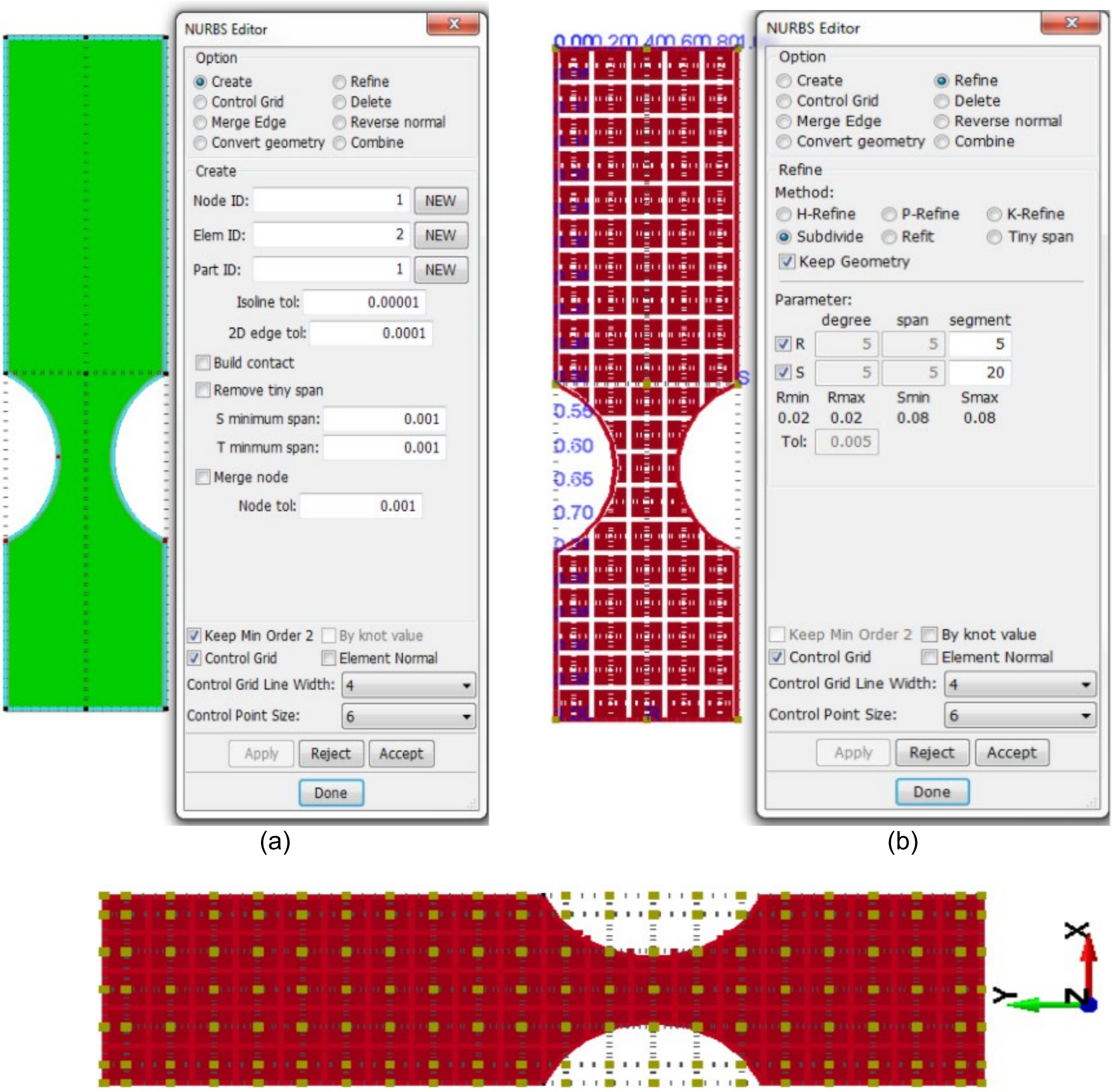


Fig.2: The mesh model construction (a) Rough mesh model; (b) Mesh refinement; (c) Final mesh model.

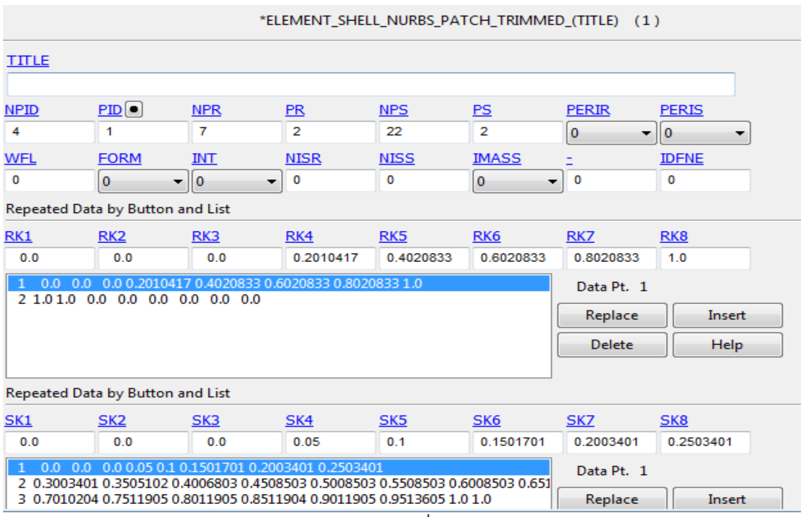


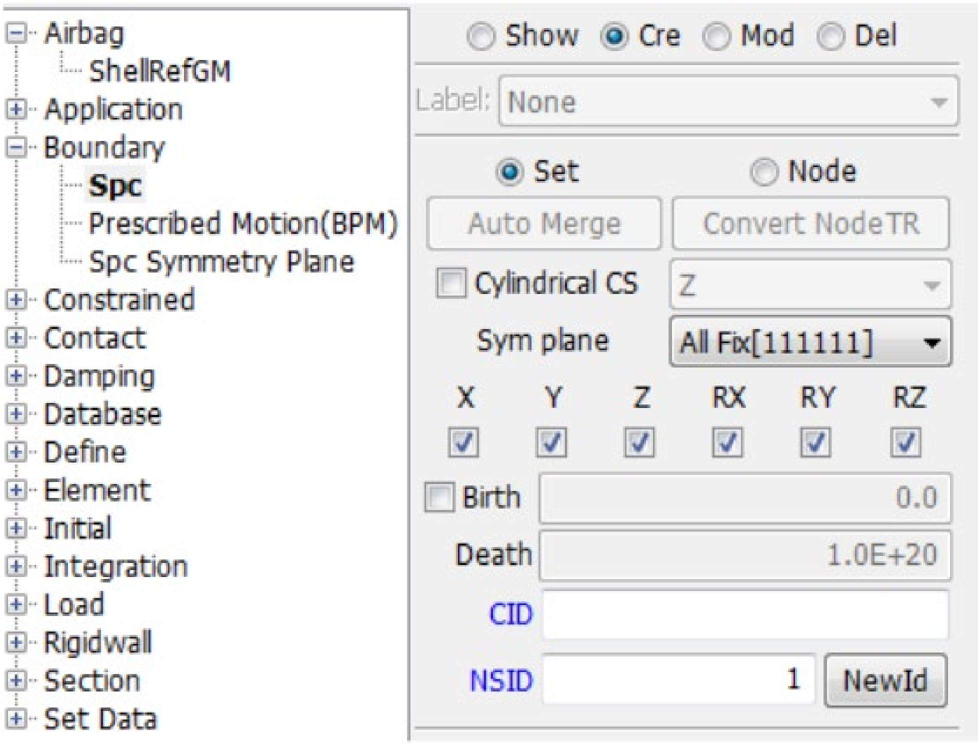
Fig.3: Element shell NURBS patch

2.3 Material, section properties and boundary condition

The elastic material properties can be defined, based on the keywords, Material_Elastic (Fig. 4 (a)), in which the Young's modulus, Poisson's ratio, and Mass density can be added. The section properties can be developed in the keyword, SECTION_SHELL (Fig. 4 (b)), in which the element formulation, the number of integration points through thickness direction, the shell thickness can be defined. Based on the keyword, BOUNDARY_SPC (Fig. 4 (c)), the boundary condition can be created on control points, in which each control point has 6 degree of freedoms that are respectively in translational and rotational constraints in local x, y, z directions.

(a)

(b)



(c)

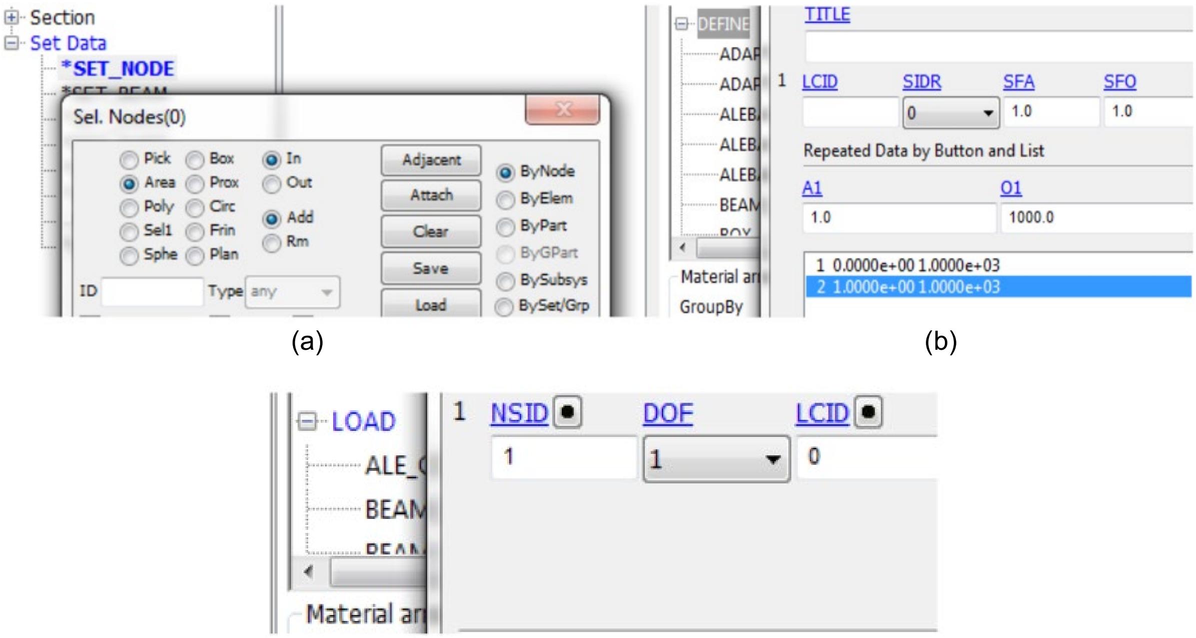
Fig. 4: Material, section properties and boundary condition setting (a) Material properties; (b) Section properties; (c) Boundary condition.

2.4 Static analysis

During isogeometric static analysis, a load can be applied directly on control points, and also on NURBS elements. The two cases would be explained in the following respectively.

2.4.1 Case 1: load applied on control points

In the first step, the control points that would be carried on loads can be selected by the keyword, SET DATE SET_NODE (Fig. 5 (a)). In the second step, the applied load value can be defined using the keyword, DEFINE_CURVE, in which the first column A1 and second column O1 respectively represent the time and load amplitude (Fig. 5 (b)). In the final step, the defined load can be applied on the selected control points though the keyword, LOAD (Fig. 5 (c)), in which the load can be applied in different directions through the selection of Degree of Freedom (DOF).



(a) (b)

(c)

Fig. 5: The setting procedures of load application on control points

2.4.2 Case 2: load applied on NURBS elements

A load can also be applied on NURBS elements, using the keyword, Load NURBS_SHELL (Fig. 6) in which the NURBS element that would receive loads can be selected using the different knot values of the knot vectors, based on the element shell NURBS patch information (Fig. 3). The applied load type can be set based on the keyword, LTYPE, and the definition of applied load though the setting of the curve is the same as previous part.

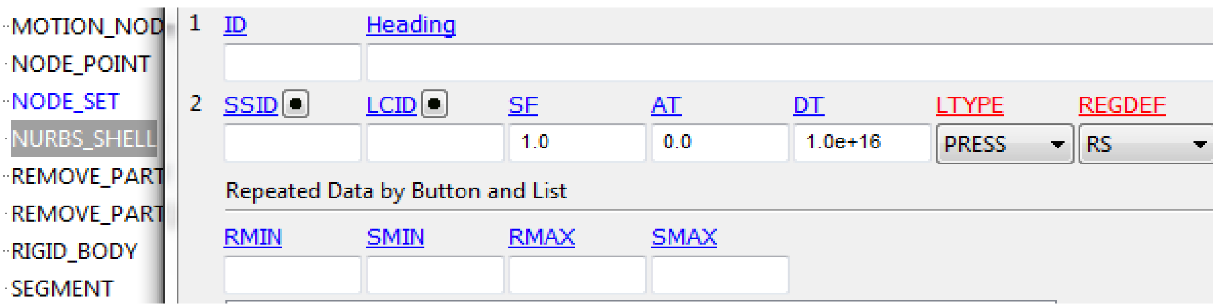


Fig. 6: Load applied on NURBS elements

2.5 Dynamic analysis

The setting process in modal analysis and fatigue analysis will be presented.

2.5.1 Modal analysis

In the first step, the number of eigenvalues to extract can be set, based on the keyword, IMPLICIT_EIGENVALUE (Fig. 7 (a)), which would be used to activate eigenvalue analysis to compute eigen modes, which would be saved in a binary database "d3eigv"; the analysis method can be selected, using the keyword IMPLICIT_GENERAL(Fig. 7 (b)), in which the time step for implicit analysis can also be defined; the analysis solution for implicit analysis can be chosen by the keyword, IMPLICIT_SOLUTION, (Fig. 7 (c)).

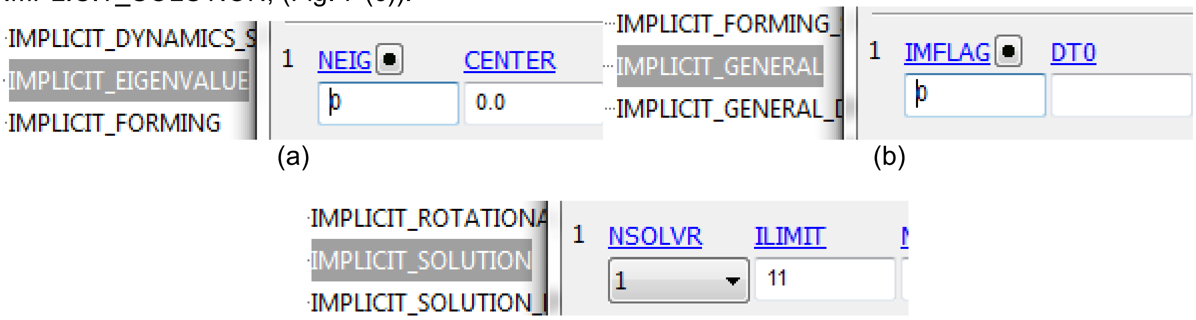


Fig. 7: Modal analysis procedures

2.5.2 Random vibration fatigue analysis

The definition of random vibration fatigue analysis in Ls Dyna can be broadly divided into two main steps. In the first step, using the keyword, FREQUENCY_ DOMAIN RANDOM VIBRATION FATIGUE (Fig. 8 (a)), some parameters which are related to random vibration fatigue analysis, including damping ratio, the excitation load type and exposure time, method for modal response analysis and random fatigue analysis can be respectively defined, also the S-N and excitation load curve, which are defined before by the keyword, DEFINE_CURVE, can be respectively selected based on the keywords, LDPSD and LCID (Fig. 8 (b)). In the second step, the post-processing keywords such as FREQUENCY_BINARY D3FTG, D3PSD, D3RMS can be defined, and the minimum, maximum response frequencies, and the output PSD number can be defined in the keyword, FREQUENCY_BINARY D3PSD, (Fig. 8 (c)).

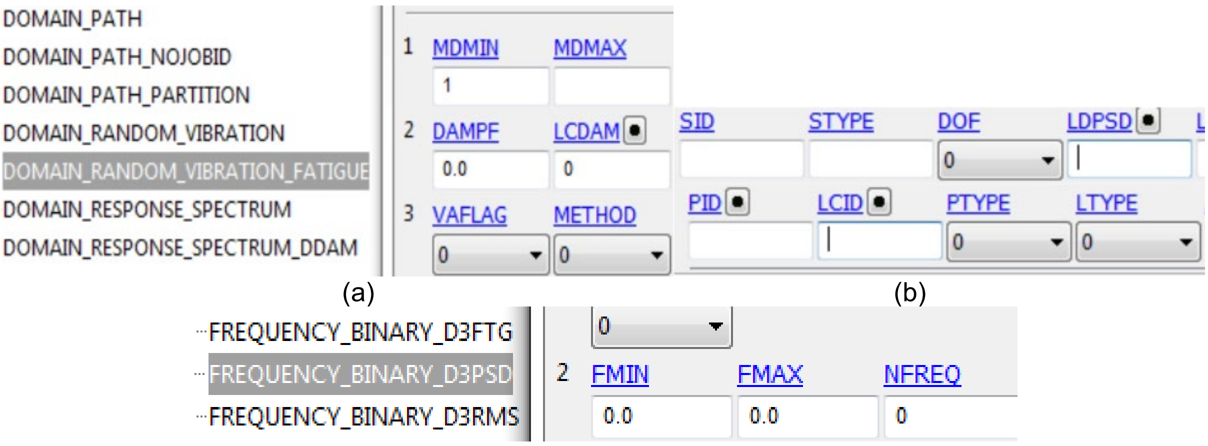


Fig. 8: Fatigue analysis procedures

3 The first case: the IGA and FEA on a notched plate model

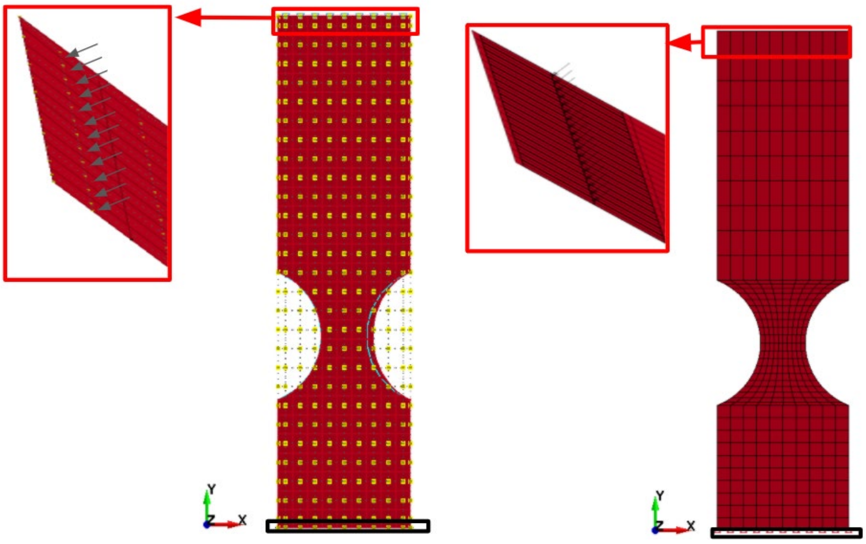
3.1 Analysis preparation

In this section, the isogeometric and finite element analyses were developed on a steel specimen with a reduced section. The mechanical properties of the material are given in Tab. 1. This plate loaded in bending is firmly clamped on one side of the beam and kept free on the other. The reduced section is used to localize the stresses away from the clamp. A random acceleration was applied to the clamping fixture. To assess the life duration, Dirlik's approach was employed in IGA and FEA.

Table 1: Material properties.

Mass density	Young's modulus	Poisson ratio
7850 kg/m³	1.7e+11 Pa	0.3

The isogeometric and finite element mesh models are presented in Fig. 9. The boundary conditions, same for IGA and FEA, are marked by the black bracket (clamp). The pressure fixed at 100 Pa and represented by red bracket, was applied on the top row elements of the models. For IGA, the load was applied on each NURBS element with a keyword LOAD NURBS_Shell in Ls Dyna.

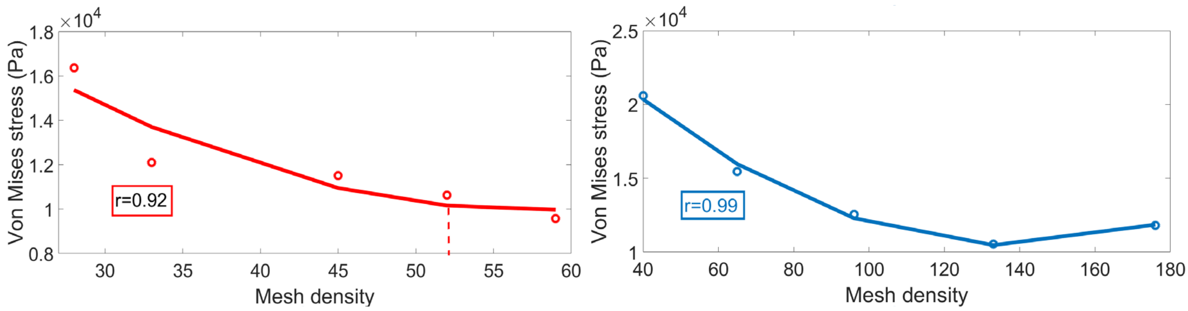


(a) (b)

Fig. 9: Mesh models and boundary conditions (a) IGA; (b) FEA

3.2 Analysis results

The von Mises stress results were used for the convergence analyses. From the analyses, presented in Fig. 10, the number of control points for IGA was chosen at 52 and the number of elements for FEA at 133.



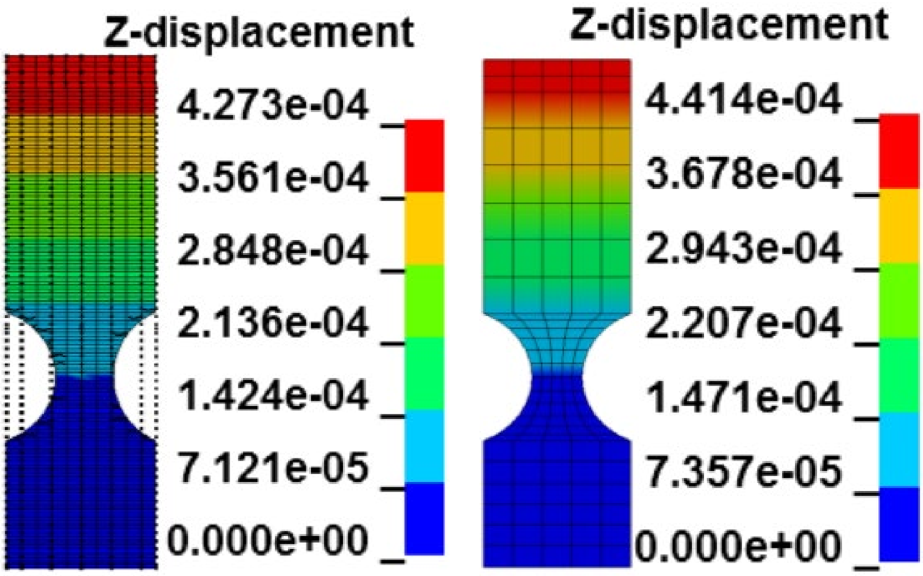
(a) (b)

Fig. 10: Convergence results and the related correlation coefficient r of the fitting curves for (a) IGA (b) FEA.

The isogeometric and finite element convergence analyses lead to similar maximum stress values (respectively 1.062e+4 and 1.054e+4 Pa) with a relative error of 0.75%. Moreover, these maxima were situated at similar locations of the reduced section. The CPU time of the analysis, respectively 231 and 462 seconds, shows that IGA is more time efficient compared with FEA.
The isogeometric and finite element random vibration fatigue analysis were developed on the obtained mesh models from convergence points. Tab. 2 and Fig. 11 respectively show the first five natural frequencies and the first vibration mode. Through comparison, it can be found that the natural frequencies and vibration mode computed from IGA and FEA have a good agreement.

Table 2: The first five natural frequencies..

methods	**1**	**2**	**3**	**4**	**5**
IGA	211	1282	1510	1623	4253
FEA	209	1141	1506	1579	4266



(a) (b)

Fig. 11: The first vibration mode (a) IGA; (b) FEA.

The random analysis was performed with LS DYNA. The acceleration PSD depicted in Fig. 12 was used to excite the specimen. This PSD was applied on the element nodes (control points for IGA) selected in boundary conditions in the z-direction to simulate the base acceleration. The damping ratio and exposure time were respectively set to 0.016 and 18000 seconds. The random vibration fatigue analysis was developed in LS DYNA to obtain the effective stress PSD and the RMS, and the expected damage. The material constants σ_f and b of the Basquin's are respectively 612.3 $\pm$ 5% MPa and -0.105, which were obtained from CES EduPack™. The margin on σ_f was used to take into consideration the material uncertainty.

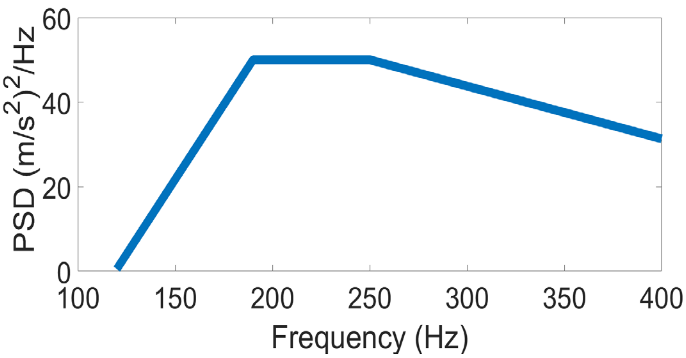


Fig. 12: Applied acceleration PSD for model 2

Fig.13 show the obtained isogeometric and finite element effective stress PSD. It can be seen that only the first natural frequency was excited by the acceleration PSD, and the resulting PSD and RMS are similar. The RMS values of the maximum effective stress obtained from IGA and FEA are respectively 5.240e+7 and 5.278e+7 Pa, leading to a relative error of -0.7%. From Fig. 14 it can be seen that the resulting expected damages are respectively 1.45 and 1.47, leading to a relative error of 1.2%. The maximum values of the damage are located on similar locations. The expected fatigue lives are reported in tab. 3. It can be observed that under different σ_f, the isogeometric and finite element fatigue life have a good agreement.

Table 3: The fatigue life results related to the 5% variation of σ_f variation (in the units of hours: minutes: seconds).

σ_f (Mpa)	581.685	612.3	642.915
IGA	2:19:40	3:27:02	4:48:43
FEA	2:17:06	3:24:30	4:46:48

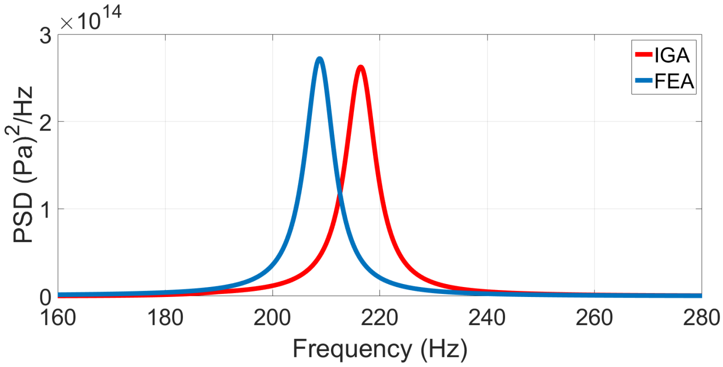


Fig. 13: Effective stress PSD.

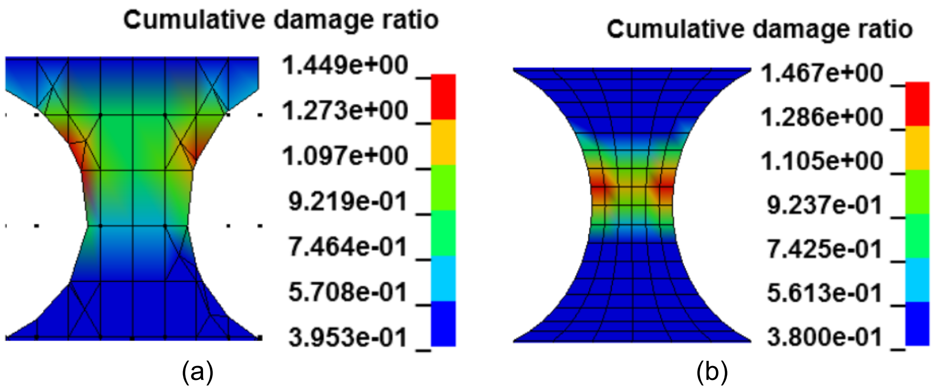


Fig. 14: The cumulative damage ratio (a) IGA; (b) FEA.

4 The second case: IGA and FEA on a wind turbine tower model

In this section, isogeometric and finite element random vibration fatigue analysis are developed on a wind turbine tower model constructed based on the reference [22].

4.1 The analysis preparations

As shown in fig. 15, the wind turbine tower model is assembled by a series of different thickness cylinders and conical shell sections, in which the geometry parameters like the height, thickness, etc are displayed in the form of mm. The tower model consists of 3 flange connections, whose base, middle and top flange thicknesses are respectively 300, 200 and 200 mm. The material properties are the same as the ones of the plate model, shown in tab. 1.

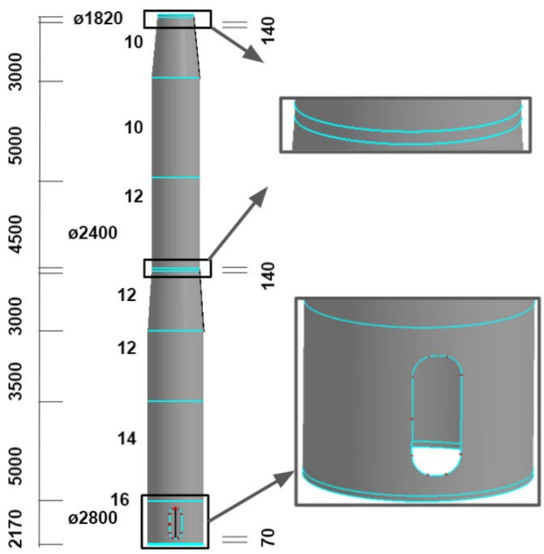
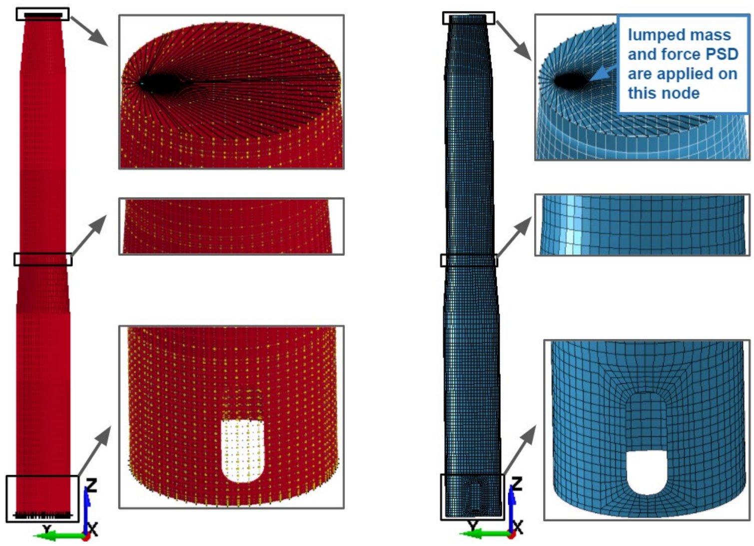


Fig. 15: Geometry model of the tower.

The isogeometric and finite element mesh models are presented in fig. 16, in which the number of control points and nodes are respectively 7639 and 12969. The finite element mesh model is created by quadrilateral 4 nodes mesh elements, and the shell element formulation of Hughes-Liu is chosen to develop fatigue analysis. For IGA, we used the isogeometric NURBS element, and also adopted Hughes-Liu shell formulation; the polynomial order of univariate shape functions in s and r-directions in the parametric space are respectively 2, and in LS-DYNA, the mesh refinement method, SUBDIVISION, is used to create more isogeometric mesh elements. After mesh generation on each section, the keyword, NODE DUPLICATION, is used to merge control points (nodes for FEA) to assemble the different sections.
To simulate the weight effects of blades, turbines, and other parts on the top of the wind turbine tower, at the height of Z = 26460 and X= -750, Y = 0 mm, a node is created to substitute the concentrated mass element of 1.002e+07 g. Then the node is connected with all control points of the top flange edge, and the weight direction is set to in negative z-direction. During analysis, the base flange of the tower model is clamped in the translational and rotational local x, y, z-directions, the vibration time and damping coefficient are set as 600 seconds and 0.01 respectively.
There are several PSD formulation for the wind turbulence, such as Kaimal Von Karman, Davonport, and Eurocode 1 spectrum. In this studying, we applied Eurocode 1 longitudinal, lateral and vertical force PSDs in x, y, z directions on the top of the tower model to develop random vibration fatigue analysis.



(a) (b)

Fig. 16: Mesh models (a) IGA; (b) FEA.

4.2 The analysis results

4.2.1 Modal analysis results: the first five natural frequencies and vibration mode

Tab. 4 and fig. 17 respectively show the first five natural frequencies and corresponding first vibration modes obtained from IGA and FEA, from which it can be observed that the frequencies and the vibration modes have a good agreement.

Table 4: The first five natural frequencies..

methods	1	2	3	4	5
IGA	2.17	2.20	13.92	13.94	15.88
FEA	2.17	2.20	13.87	13.88	15.73

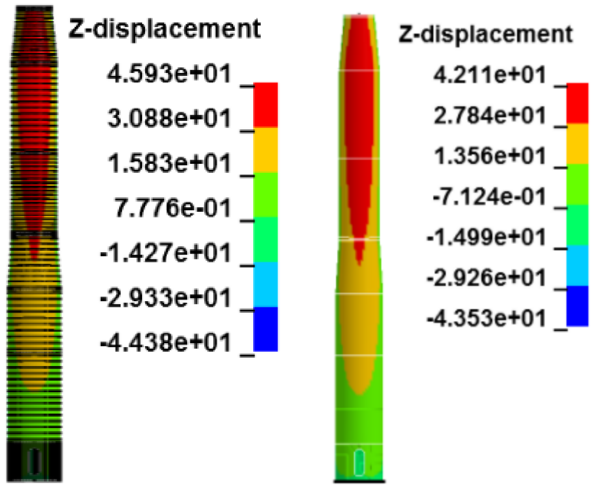


Fig. 17: The first vibration mode (a) IGA; (b) FEA.

4.2.2 Fatigue analysis results: effective stress PSD, RMS and cumulative damage ratio

Fig. 19 and 20 show the calculated isogeometric and finite element effective stress PSD and RMS, in which only the first natural frequency is excited by the applied force PSD. It is observed that isogeometric and finite element PSD and RMS display a good agreement, in which the maximum effective stress RMS from IGA and FEA is 16.18 and 15.78 pa respectively, leading to the relative error of 2.53%. From fig. 21, it can be seen that the obtained isogeometric and finite element cumulative damage ratios are respectively 4.030e-5 and 4.031e-5, and the maximum damage ratios are located on similar elements close to the door edge. The expected isogeometric and finite element fatigue life E[T_f] are 1.4888e+07, and 1.4885e+07 seconds respectively. Based on the Matlab program, the isogeometric and finite element damage ratios are respectively 4.022e-5 and 4.032e-5, which are in a good accordance with the damage ratios computed from Ls Dyna.

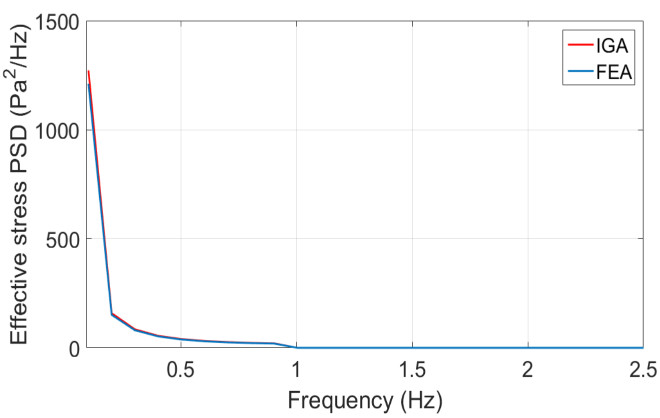
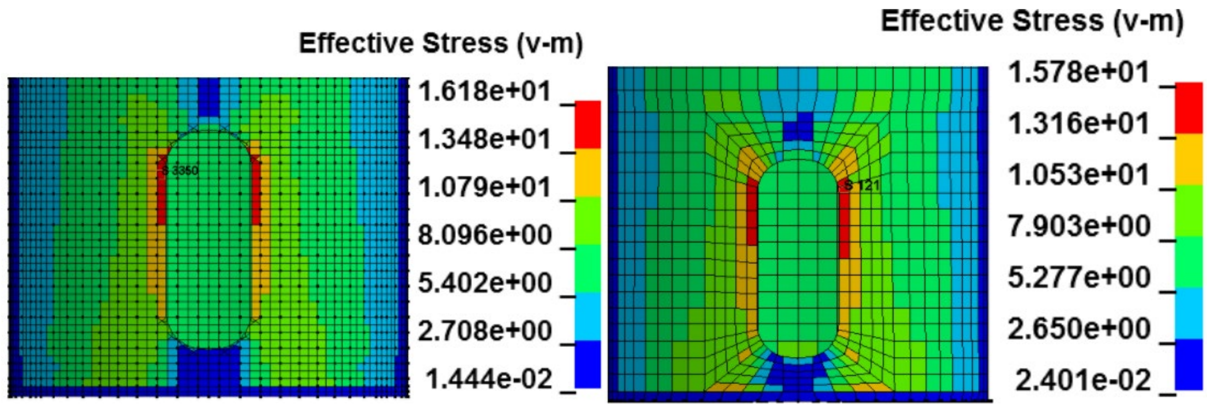
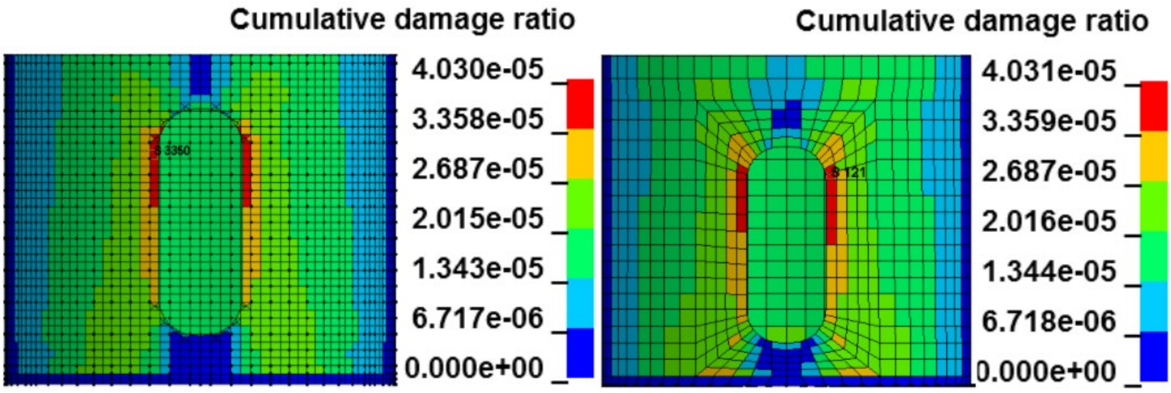


Fig. 18: The effective stress PSD.



(a) (b)

Fig. 19: The effective stress RMS (a) IGA; (b) FEA.



(a) (b)

Fig. 20: The cumulative damage ratio (a) IGA; (b) FEA.

5 Summary

In this studying, we considered convergence analysis and random vibration fatigue analysis on a plate and tower model using IGA and FEA, in which the isogeometric and finite element damage results were validated by the Matlab program.
For the plate model, the specimen is clamped from one side, and the pressure load and random acceleration in a vertical direction to the plate surface are applied to develop convergence and fatigue analyses. From convergence analysis, it was shown that the IGA and FEA can lead to similar maximum stress values with a relative error of 0.75%, and are situated at similar locations of the reduced section. The CPU time of the analysis, respectively 231 and 462 seconds, shows that IGA is more efficient compared with FEA.
For the tower model, during the analysis, the model is clamped on the base flange, and random force PSD in a vertical direction to the tower surface is applied to the concentrated mass element.
From modal analysis, it can be found that the obtained first five natural frequencies and vibration modes from IGA and FEA have a good agreement. Fatigue analyses show that the obtained isogeometric and finite element maximum effective stress RMS are 16.18 and 15.78 pa with a relative error of 2.53%, and cumulative damage ratios are 4.030e-5 and 4.031e-5, which have a good agreement. Based on the Matlab program, the isogeometric and finite element damage ratio are respectively 4.022e-5 and 4.032e-5.
On the other hand, in the aspect of the mesh refinement process, for IGA, it is not necessary to create mesh elements on the original geometry model. It is sufficient to develop mesh elements on the previous mesh model, and so the mesh refinement time can be largely saved. However, for the FEA, the refinement process is mandatory to communicate with the original geometric model, and so this process is more time-consuming in LS Dyna software.
In addition, IGA can predict the fatigue life using fewer NURBS elements and integration points in the thickness direction(for the plate model), which correspond very well to the fatigue life computed by FEA, with the relative errors of 0.68% (plate model) and 0.002%(tower model). Through the comparison of numerical analysis results, it can be observed that the obtained isogeometric, finite

element PSD and RMS have a good agreement, leading to conclude that IGA is suitable for the random vibration fatigue analysis.

6 Literature

[1] T.J.R. Hughes, J.A. Cottrell, Y. Bazilevs: "Isogeometric analysis: CAD, finite elements, NURBS, exact geometry and mesh refinement", Computer Methods in Applied Mechanics and Engineering, 194 (2005): 4135-4195.

[2] Lu J: “Isogeometric contact analysis: "Geometric basis and formulation for frictionless contact ", Computer Methods in Applied Mechanics and Engineering, 200 (2011): 726-741.

[3] Temizer I, Wriggers P, Hughes T J R: "Contact treatment in isogeometric analysis with NURBS", Computer Methods in Applied Mechanics and Engineering, 200 (2011): 1100-1112.

[4] Temizer I, Wriggers P, Hughes T J R: "Three-dimensional mortar-based frictional contact treatment in isogeometric analysis with NURBS", Computer Methods in Applied Mechanics and Engineering, 200 (2011): 726-741.

[5] Y. Bazilevs, T.J.R. Hughes: "NURBS-based isogeometric analysis for the computation of flows about rotating components", Comput Mech, 43 (2008): 143–150.

[6] Y. Bazilevs,V.M. Calo, Y. Zhang, T.J.R. Hughes: "Isogeometric Fluid–structure Interaction Analysis with Applications to Arterial Blood Flow", Comput Mech, 38 (2006): 310–322.

[7] Y. Bazilevs,V.M. Calo, Y. Zhang, T.J.R. Hughes: "Isogeometric fluid-structure interaction: theory, algorithms, and computations", Comput Mech, 43 (2008): 3–37.

[8] Xiaoping Qian: "Full analytical sensitivities in NURBS based isogeometric shape optimization", Computer Methods in Applied Mechanics and Engineering, 199 (2010): 2059-2071.

[9] Wolfgang A. Wall, Moritz A. Frenzel, Christian Cyron: "Isogeometric structural shape optimization", Computer Methods in Applied Mechanics and Engineering, 197 (2008): 2976-2988.

[10] B. Hassani, S.M. Tavakkoli, N.Z. Moghadam: "Application of isogeometric analysis in structural shape optimization", Scientia Iranica, 18 (2011): 846-852.

[11] S. Shojaee and N. Valizadeh and M. Arjomand: "Isogeometric structural shape optimization using particle swarm algorithm", Iran University of Science & Technology, 1 (2011): 633-645.

[12] J. Kiendl, K.-U. Bletzinger, J. Linhard, R. Wüchner: "Isogeometric shell analysis with Kirchhoff–Love elements", Computer Methods in Applied Mechanics and Engineering, 198 (2009): 3902-3914.

[13] D.J. Benson, Y. Bazilevs, M.C. Hsu, T.J.R. Hughes: "Isogeometric shell analysis: The Reissner–Mindlin shell", Computer Methods in Applied Mechanics and Engineering, 199 (2010): 276-289.

[14] D.J. Benson, Y. Bazilevs, M.C. Hsu, T.J.R. Hughes: "A large deformation, rotation-free, isogeometric shell". Computer Methods in Applied Mechanics and Engineering, 200 (2011): 1367-1378.

[15] Y. D.J. Benson, Y. Bazilevs, M.C. Hsu, T.J.R. Hughes: "T-spline finite element method for the analysis of shell structures", International Journal for Numerical Methods in Engineering, 80 (2009): 507-536.

[16] Sang Jin Lee, Kyoung Sub Park,s: "Vibrations of Timoshenko beams with isogeometric approach", Applied Mathematical Modelling, 37 (2013): 9174-9190.

[17] Weeger, O., Wever, U. & Simeon, B.: "Isogeometric analysis of nonlinear Euler–Bernoulli beam vibrations", Applied Mathematical Modelling, Nonlinear Dyn, 72 (2013): 813-835.

[18] Clemens V. Verhoosel Michael A. Scott Thomas J. R. Hughes: "An isogeometric analysis approach to gradient damage models", International Journal for Numerical Methods in Engineering, 86 (2011): 115-134.

[19] Bazilevs, Y., Deng, X., Korobenko, A., Lanza di Scalea, F., Todd, M. D., and Taylor, S. G: "Isogeometric Fatigue Damage Prediction in Large-Scale Composite Structures Driven by Dynamic Sensor Data", Journal of applied mechanics, 82 (2015): 115-134.

[20] J.A. Cottrell, A. Reali, Y. Bazilevs, T.J.R. Hughes: "Isogeometric analysis of structural vibrations", Computer Methods in Applied Mechanics and Engineering, 195(2006): 5257-5296.

[21] Dongdong Wang, Wei Liu, Hanjie Zhang: "Novel higher order mass matrices for isogeometric structural vibration analysis". Computer Methods in Applied Mechanics and Engineering, 260 (2013): 92-108.

[22] N Bazeos, G.D Hatzigeorgiou, I.D Hondros, H Karamaneas, D.L Karabalis, D.E Beskos: "Static, seismic and stability analyses of a prototype wind turbine steel tower", Engineering Structures, 24 (2002): 1015 - 1025.