Science Arena Publications
Specialty Journal of Engineering and Applied Science
ISSN: 2520-5943
Available online at www.sciarena.com
2018, Vol, 4 (1): 30-48

The Investigation of Free Vibration Analysis of Bovine Tibia Bone by Using “Finite Element Method”

Hamid Dehghan Tarazjani[1], Saman Soheilizade[2*]

[1]Supervisor, Department of Mechanic Engineering, Parand Branch, Islamic Azad University,
[2]Master Of Science Mechanic Engineering, Applied design, Azad University (Parand Branch).

***Corresponding Author**

Abstract: *Nowadays, the study of bones that are compound materials featuring different characteristics in various body parts has been proposed as a novel subject in mechanic engineering and biomechanics. The present study tends to analyze the free vibrations (attainment of natural frequencies) of a specimen of cow fibula (considering the constraints on access to real human bone in Iran). At first, a 3D finite element model of the cow fibula was prepared using CT-scan images (the model is created by MIMICS software) following which the model was transferred to Abaqus Software to be further processed. In the beginning, the characteristics of the bone material is specified in the form of elastic inhomogeneous isotropic (based on density-elasticity relations offered by Carter, Keller and Morgan) (discrete model) and, then, the properties of the materials were inserted in a continuous manner for the individual bone parts following which the natural frequencies were acquired. Next, the effect of both of the models on the vibration attributes of the bone specimen was evaluated and the obtained result were compared with the laboratory results and it was made clear that the approach shift from discrete to continuous provides for obtaining more acceptable results (closeness of the answers to the experimental numbers) and it was found out that Morgan’s relations provide for laboratory results closer to the real data.*

Keywords: *Natural Frequency, Cow Fibula, Finite Element.*

INTRODUCTION

The study of the prior research and printed articles is indicative of the scarcity of researches that have compared the various past approaches with various modern approaches in a real sample. Due to the nearly impossible access to a real sample of human bones in Iran, the present study has attempted to perform all the experiments given herein using cow bone. So, the present study tries performing numerical, software-based and finite element analyses to investigate the free vibrations of the cow calf or thigh bone. The study is important in regard of the proper specification of material in a bone sample in analyses performed in such areas as mechanic concerning the tension and modal analysis because, as it is known, the bone specifications differ in various body parts (Taddi, 2007).

The followings are some examples of the studies carried out in this area:

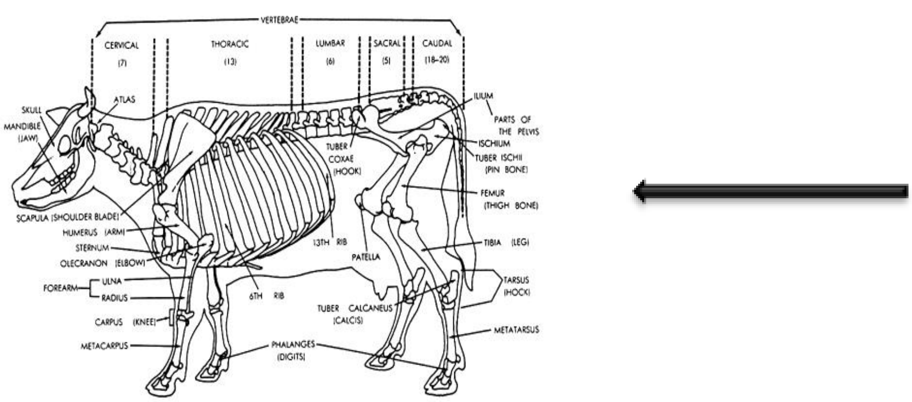
In 2007, the mechanic engineering group of Milan’s Polytechnic University offered a software type named Bone Mat that makes use of prior approaches (Carter, Keller, Morgan and others) to specify the bone sample materials in an automatic manner. It is worth mentioning that the software was edited in 2010 and Re 01 version called Bone Mat II was supplied and its validity was evaluated by some individuals like Taddi who compared the results with laboratory findings and reached a difference equal to 8.6%. It has to be pointed out that individuals like Shileo et al (from Italy) were seeking to figure out whether the proper selection of density-elasticity relation can lead to more precise predictions of yield within the format of a customized model development strategy or not? (Christian Wong et al., 2010; Suk-Hun Kim et al., 2010) They concluded that the density-elasticity relation exerts a high effect on the numerical estimation accuracy (one of the previously-tested regulations showed a good consistency between the laboratory results and numerical calculations) (Bekir Bediz, Nezat, 2010; Neugebauer et al., 2011; Taddi, 2007; Taylor, 2002). It is noteworthy that they all make use of Taylor Approach in computing the density value.

In 2013, Roger Schols et al (orthopedic surgery group in Germany's Leipzig University) dealt with the validation of density-elasticity relations to perform finite element modeling of human pelvis using modal tests. A mean Young Modulus was obtained for each element through relating the specifications of each element to the CT-scan data (considering the material being an isotropic homogeneous material) and three Young Modulus relations were defined for density that were posited respectively based on Carter, Keller and Morgan relations. It is notable that they utilized the article presented by Taylor (2002) to calculate density changes and performed modal analysis using Ansys Software so that the natural frequencies and mode forms of the bones could be determined. Then, they came to the conclusion that Morgan's density-elasticity relation can better demonstrate the match between the finite element model's natural frequency and modal test sample's natural frequency (Roger Schols, Folk Hoflma, 2013; Gupta, Tse, 2014).

Study Method:

Preparing A Sample of Cow Fibula:

Due to the difficulty of accessing real human bone in the present study, a cow fibula was used to perform modal analyses. The present study has made use of tibia of a cow the schematic view of which has been illustrated below: (Arild Aamodet et al., 2010)



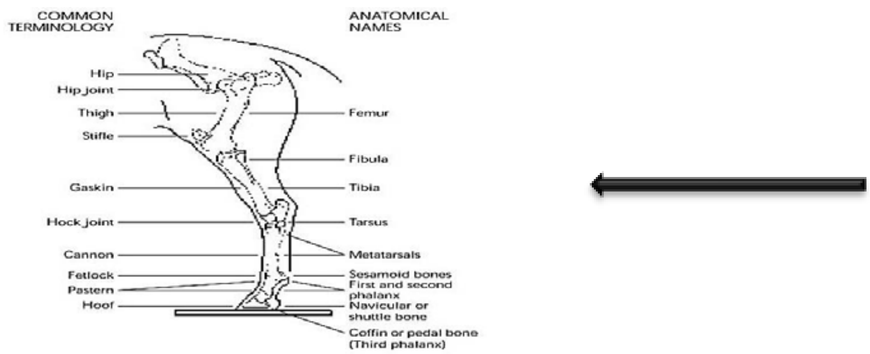


Figure 1: tibia images

Performing CT-Scan:

One-millimeter bones from the ending section (epiphysis) and middle sections (diaphysis) were prepared using 1024 × 1280 segment density (3D image measurement unit) in GE HI Speed Device in 120kv and 110MA. (Chen, Schule, 2010)

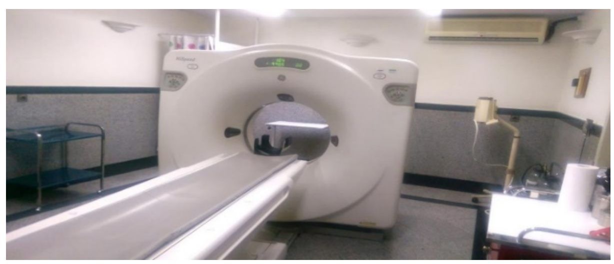


Figure 2: image pertaining to CT-Scan device

The information pertinent to the device is given in table (1). (It has to be mentioned that the corresponding device had been previously calibrated before the initiation of scanning.)

Table 1: CT-Scan device specifications

Slice thickness	0.6-10 mm	Reconstruction time	1.1 second(S) per image	Image stacks	13.5
MA range	10-440 MA	Matrixes, Pixels	1280 × 1024	Voxel Resolution	100 (mm)
Scan volume	170	Scan localizer	Incandescent light		

Figures (3) to (4) display the images pertaining to CT-Scan.

Figure 3: images pertaining to CT-Scan 1

Figure 4: images pertaining to CT-Scan 2

After performing CT-scan, the operator was asked to offer the information in DICOM output format so that they could be readily applied for further software processing works and higher quality 3D model preparation. A total of 450 DICOM output format images were obtained. (Biomechanics of hard tissues, Modeling, Testing and Material. Edited by Andreas Wager, 2010; Guindy, Anna University, Chennai, India, 2009)

Three-Dimensional Model Preparation Using MIMICS Software:

- **Stages of 3D Model Preparation:**

The following parts express the summary of the stages taken for the preparation of the 3D model:

1) CT Scan…..Prepraing Dicom Data.....Slice 1mm
2) Mimics
   2-1) Inpout Dicom Data to Mimics Software
   2-2) Information Categorization,Gray Scale,Gray Value
   2-3) Segmentation,Thresholding
   2-4) Preparing Mask,Initial 3D Model and Multiple Slice Edit
3) STL Output and Input to 3 Matic to meshing and 3D Advanced Design
   3-1) Triangle Edge and Optimizing,Smooth
   3-2) Uniform Mesh,Remeshing
   3-3) Create Volume Mesh
   3-4) Calculating the Node and Volume Element
   3-5) Histogram

- **MIMICS Software:**

MIMICS (materialise's interactive medical image control system) is a software type for processing medical images and creation of 3D models. The software employs 2D medical cross-sectional images like computed tomography and magnetic resonance imaging to create 3D models that can be directly applied for fast sampling, CAD modeling, surgical simulations and engineering analysis[1].

- **MIMICS Software Implementation:**

At first, a 3D model of tibia is prepared in Mimics using CT-Scan images taken for cow tibia.

1) CT Images' Insertion: to do so, the option "New Project Wizard" is clicked in "File" Menu and the images are inserted into the software.

[1] Mimics Student Edition Course Book(Mimics SE) from WWW.MATERIALISE.COM

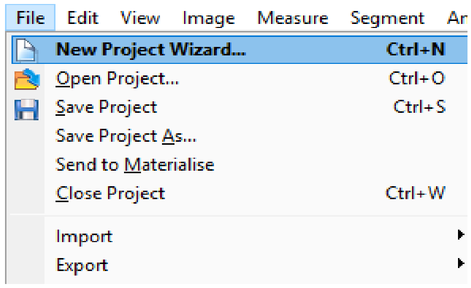


Figure 5: New Project Wizard Menu

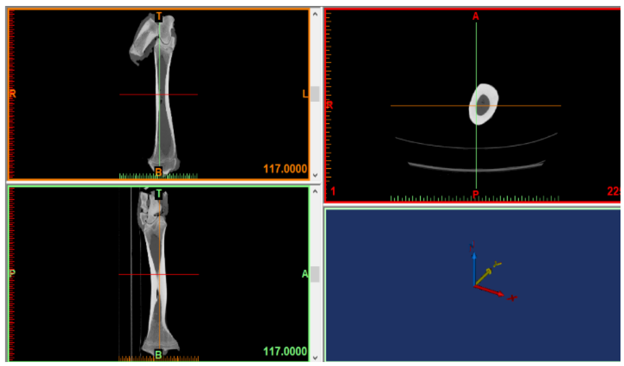


Figure 6: image of the bone specimen in MIMICS Software

2) In the next stage, a mask can be made through "Segmentation" menu and selection of the option "Threshold" and setting thresholding domain on "bone" following which the bone data can be extracted from the images.

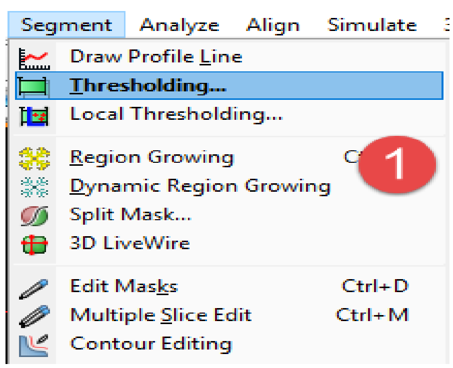


Figure 7: Mask Menu

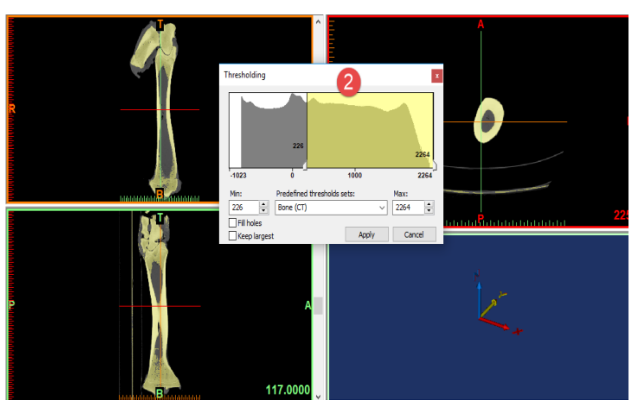


Figure 8: the method of creating Mask in MIMICS Software

3) After defining the threshold, the option "Calculate 3D from Mask" can be used to make a 3D model of what has been realized by the software as bone.

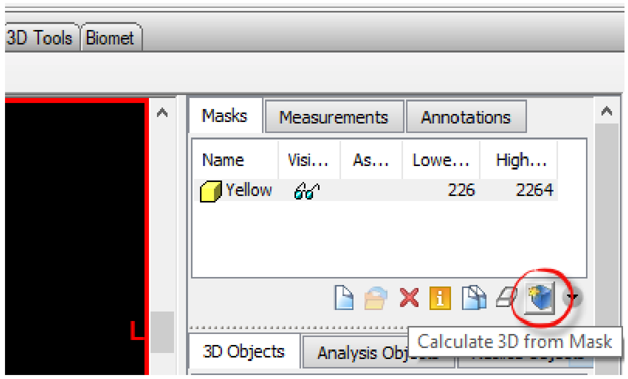


Figure 9: preparation of 3D mask

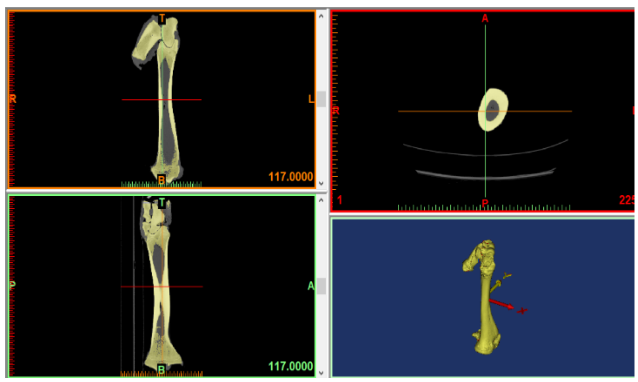


Figure 10: the created 3D model

The preliminary model takes the form shown in figure (10). But, it has to be adjusted and the redundant parts have to be cleaned.

4) To perform the corrections, the option "Multiple Slice Edit" is selected from "Segmentation" Menu.

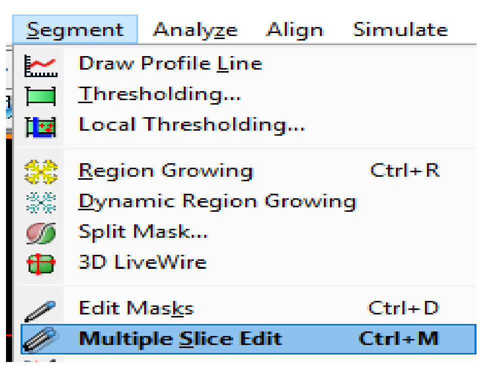


Figure 11: Multiple Slice Edit menu

In Multiple Slice Edit Menu, the page of interest for the performing of the corrections is chosen and the operator is set on "Remove". Now, the extra parts in each slice of Mask can be eliminated so that the intended model could be attained. In every elimination of the extra parts, the button "Apply" should be clicked so that the changes can take effect.

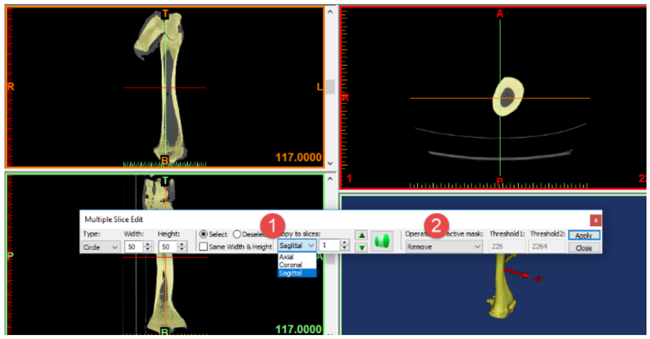


Figure 12: removing the extra parts using "Remove Button"

5) After the intended corrections were done on the Mask, a new model can be again created via clicking on Calculate 3D Form Mask.

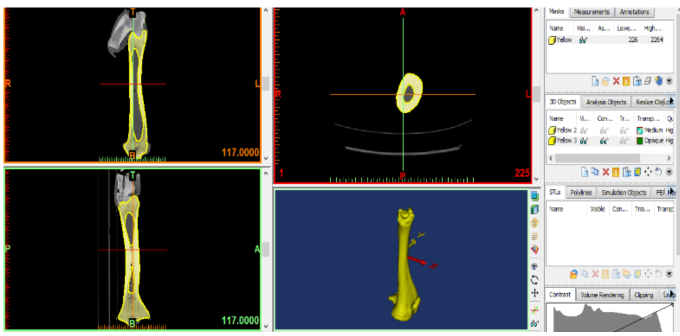


Figure 13: the final 3D model

In the end, the 3D model created in MIMICS Software can be exported in the desired format.

6) The model created in MIMICS is imported into 3-matic software.

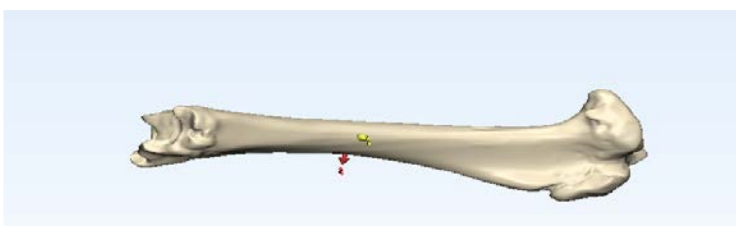


Figure 14: the output 3D model

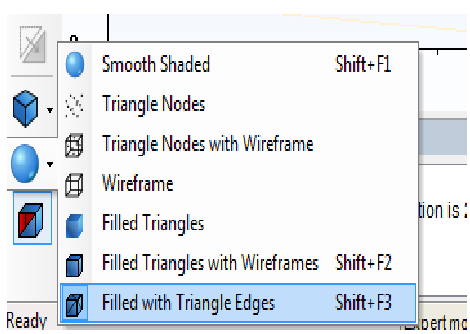


Figure 15: Triangle Edge Menu

In the next section, the created mesh will be smoothed so as to make it fitting the upcoming analysis.

Figure 16: Smooth Menu in 3 Matic

As it is observed, the software has a "smooth" part in its upper toolbar in the "fix" key through the selecting of which followed by the selecting of the model, the model surface can be processed.

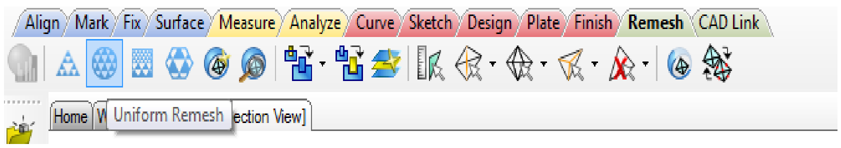


Figure 17: Uniform Remesh Menu

Then "Remesh" key is used to choose "Uniform Remesh" so as to render the model surface meshing uniform.

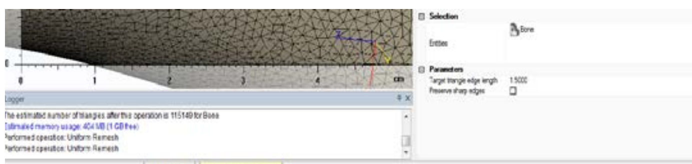


Figure 18: the uniform surface created

It can be seen in figure (18) that the length of the triangles' edge has been set on 1.5mm for the model and the number of the estimated elements before the implementation of the command is observable in the "Logger" part of the software.

After this, an internal meshing has to also be made for the model. To do so, the following action is taken that "Create Volume Mesh" is chosen from the "Remesh Key".

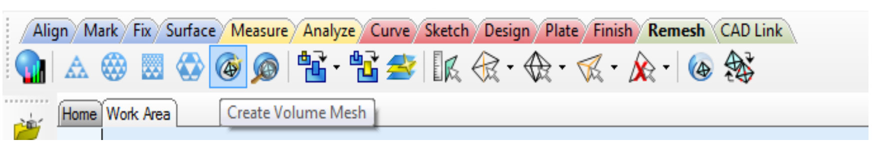


Figure 19: creation of model using "Create Volume Mesh"

Then, as shown in figure (20), the element type is selected of tet4 type and the maximum edge length is set on 3mm following which "apply" button is pressed in order for the model to undergo meshing. This stage of the work might take a while.

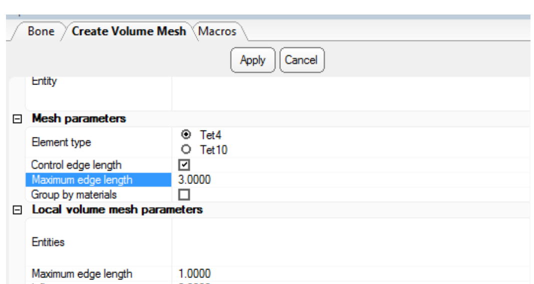
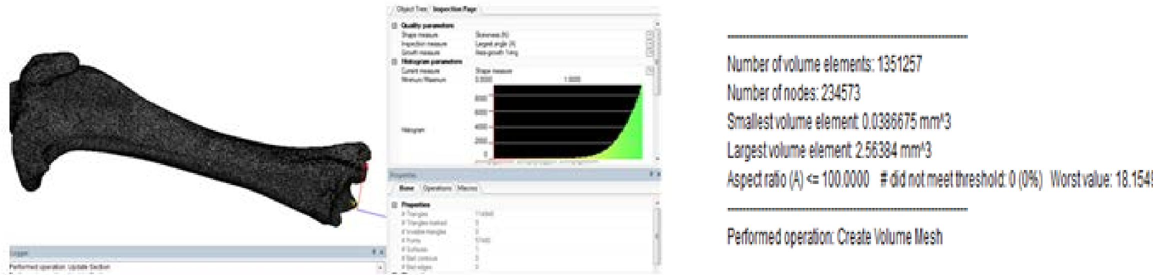


Figure 20: meshing accomplished

It can be observed in the end of the work that how many volume elements and nodes have been created for the figure and the other information can also be found in table (3).

Table 3: the number of nodes and elements of the model



Histogram Menu has been depicted in the above right-hand side image. Meshing is situated in the green region. The 3D meshed model is ready to be analyzed in Abaqus Software.

Calculation of Natural Frequency (Discrete Method):

- **Using Abaqus Software (with only one material):**

After the entire 3D stages were finished, the 3D (meshed) model is inserted into Abaqus Software (with no material specification). An Abaqus applicable output with INP extension or the very Abaqus extension output is taken from MIMICS software so that it could be implemented in Abaqus. (Abaqus Software, Help, 2017)

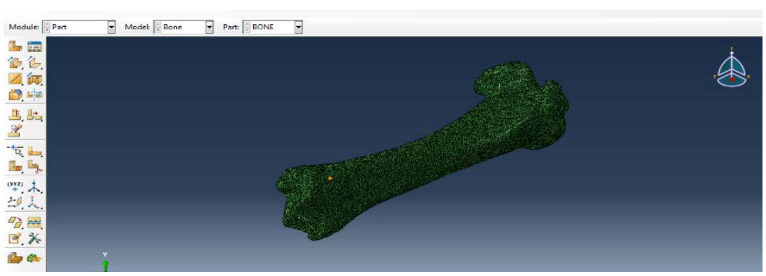


Figure 21: MIMICS Output in Abaqus Software

- **Material Specification:**

In the next stage, it is required to specify the materials. So, the information is inserted into the software corresponding to the previous articles (the values have been given below) (Amaziah Walter Otunyo et al., 2014):
Elasticity Module: 18.7 Gpa
Poisson Coefficient: 0.3
Density: 1670 kg/m3

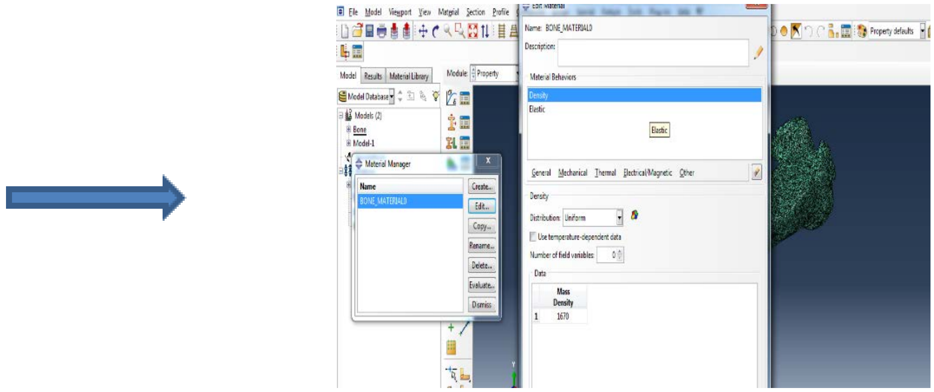


Figure 22: density value specifications

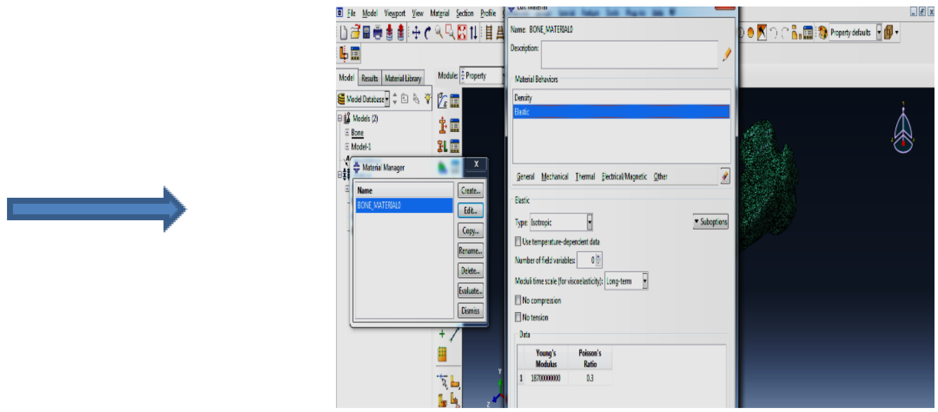


Figure 23: elasticity module specifications

It has to be mentioned that the entire dimensions should be based on SI system. According to table (4):

Table 4: ISI dimensions

Quantity	SI	SI(mm)	SI	US Unit(ft)	US Unit(inch)
Length	m	mm	m	ft	in
Force	N	N	kN	lbf	lbf
Mass	kg	$tonne\ (10^3\ kg)$	$tonne$	$slug$	$lbf\ s^2/in$
Time	s	s	s	s	s
Stress	$Pu\ (N/m^2)$	$MPu\ (N/mm^2)$	kPu	lbf/ft^2	$psi\ (lbf/in^2)$
Energy	J	$mJ(10^{-3}J)$	KJ	$ftlbf$	$inlbf$
Density	$kg/\ m^3$	$tonne/mm^3$	$tonne/m^3$	$slug/ft^3$	$lbf\ s^2/in^4$

- **Boundary Conditions:**

In the next part, the type of the analysis intended to be executed on the model is selected. In the case of the present study, modal vibration analysis is the evaluation of choice and the conditions are set on Free-Free.

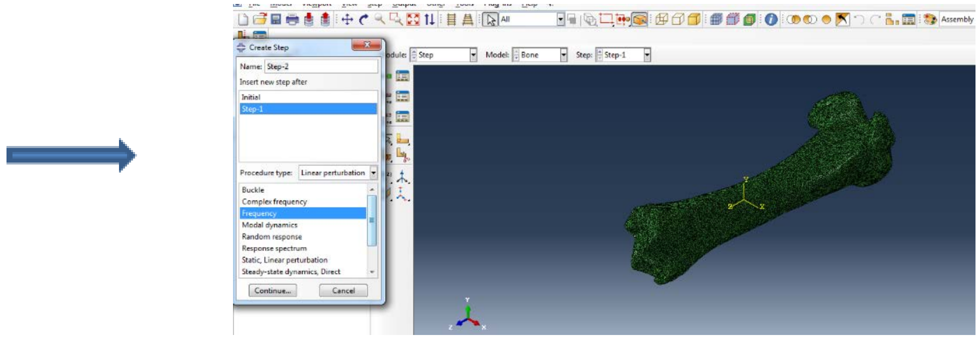


Figure 24: implementation of Free-Free Boundary Conditions

- **Natural Frequencies:**

In this section, all of the vibration modes that are intended to be exhibited in the output are inserted. The default options are accepted for the rest of the settings. (Abaqus Software, Help, 2017)

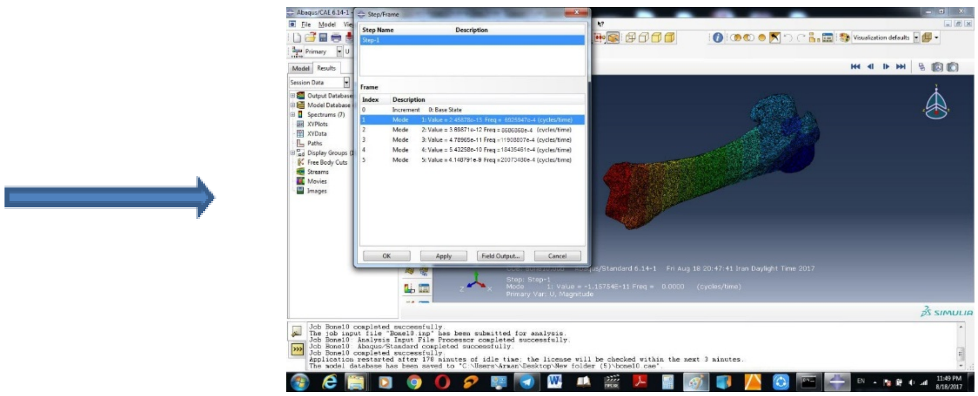


Figure 25: natural frequency output

The value outputs are given in table (5) more clearly.

Table 5: natural frequency values

Frequency	F1	F2	F3	F4	F5
No.	692	868	1190	1843	2007

Calculation of Natural Frequency Via Abaqus Software Using 42 Different Groups:

In the next stage, Abaqus-fitting mesh output is taken from the intended file in 3D-matic software so that the material of interest could be specified. (Arild Aamodet et al., 2010)

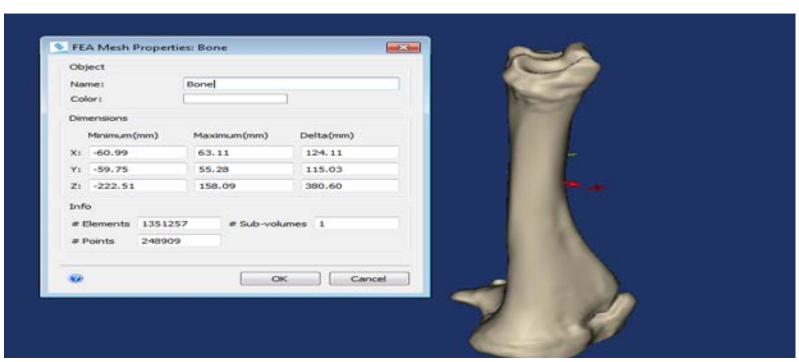


Figure 26: geometrical properties of the bone specimen

In figure (26), the material is yet to be specified for the intended model. The complete model specifications were observed before material specification. According to the following menu, the option "Assign Material" is selected that is used to specify the material in the software.

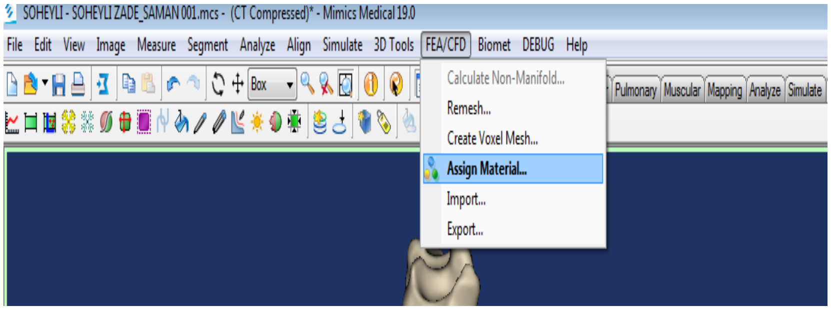


Figure 27: assign material menu

The intended page is opened in MIMICS software and "Gray Value Based" approach is utilized to perform specification (because the model has been created based on CT-Scan images). It is evident in the image that the minimum Hounsfield value, that is a linear transformation of ray attenuation coefficient in CT-Scan indicating the density of the tissue in each part of the image; the attenuation coefficient of the water ray is assumed equal to zero and that of the air is assumed 100, is -1018 and maximum Hounsfield. (Khan, Warkhedekar, 2010) value is +1970.

- **Material Specification:**

The various regions of the bone feature different density and elasticity modules that have to be specified. According to the formula at hand for bone density that was excerpted from the article and approach proposed by Taylor (2002), the density part is filled following which the elasticity module is specified. Elasticity module is a function of density. In this stage, Carter and Hayes approach (1977) is applied (Roger Schols, Folk Hoflma, 2013; P. Schrock A, B, , 2015; Taylor, 2002):

$$\rho_{eff} = AHU + 1000 (kg/m^3)$$
$$E = a\,\rho^b$$

In Carter and Hayes approach, a is equal to 3.79 and b is 3.
In the end, a fixed Poisson ratio equal to 0.3 is inserted for all of the parts. Upon the insertion of these values, the software corrects the histograms related to the materials so that they could match to the inserted numbers. For example, the segmentations for the 42 groups have been shown in figures (29) to (33).

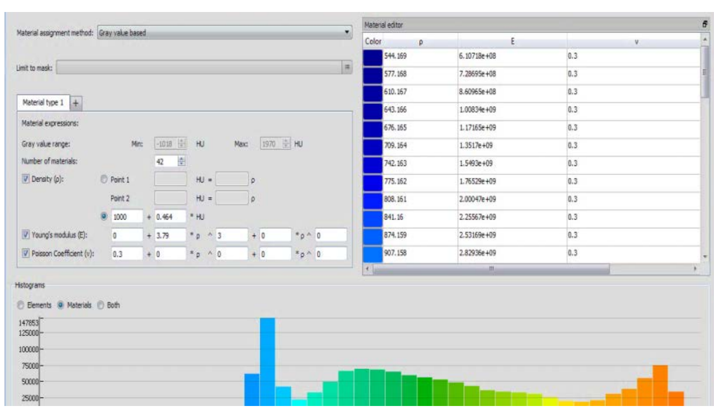


Figure 29: specifications for 42 different groups

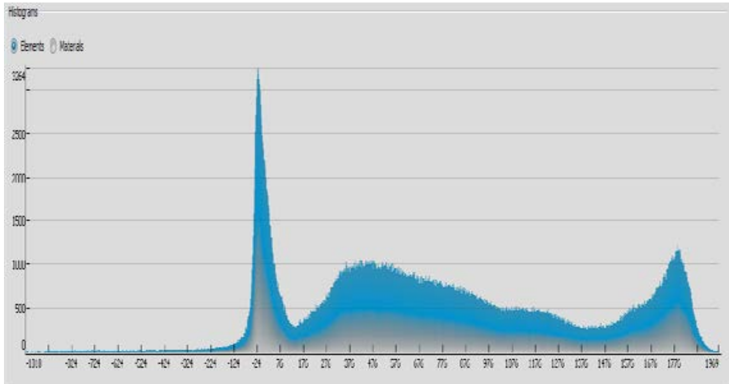


Figure 30: the range of the specified numbers 1

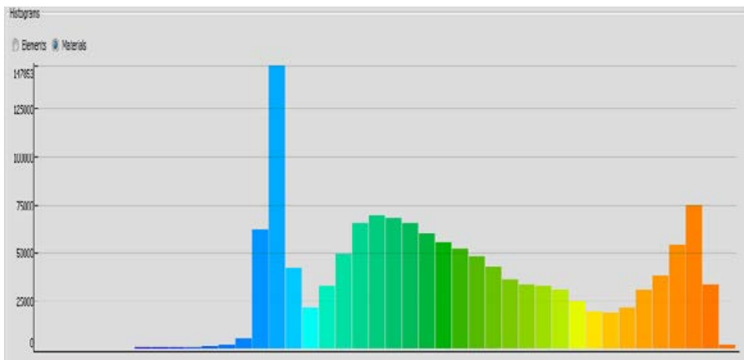


Figure 31: the range of the specified numbers 2

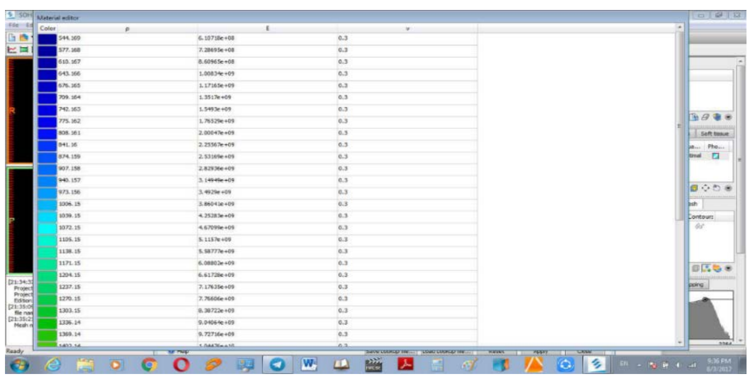


Figure 32: density and elasticity module 1

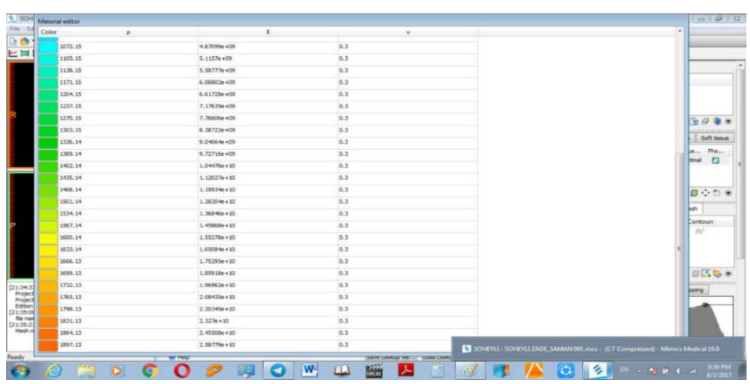


Figure 33: density and elasticity module 2

In figures (32) and (33), the values related to density and elasticity module and Poisson coefficient for the 42 groups have been displayed. It is worth mentioning that the software automatically carries out the calculations. (Khan, Warkhedekar, 2010; Helgason et al., 2008).

Calculation of Natural Frequency Using Continuous Distribution:

One of the most important constraints in obtaining the exact value of the natural frequency is the limited number of the materials constituting the bone and their discontinuous (discrete) distribution. Up to this part, the bone density span was segmented into 42 parts and a mean value was utilized as the density of the entire part in the calculations. The use of MIMICS software for determining the material distribution has always been accompanied by such a constraint. However, the number of the used materials can be increased as desired but this might result in an increase in the work volume. The objective in continuous distribution is the assignment of materials to every bone spot so that the discreteness problem could be resolved following which the model would feature continuous material specification before its natural frequency is obtained.

- **Abaqus Program Execution:**

The following steps should be taken before linking Abaqus to Matlab:

1) The mesh-free geometry is inserted into Abaqus.
2) Meshing is conducted in Auto Mesh Format. In figures (34) and (35), a MIMICS output specimen in Abaqus, a meshing sample and a geometrical specification output specimen can be seen (due to the spatial limitations, all of the images have not been shown).

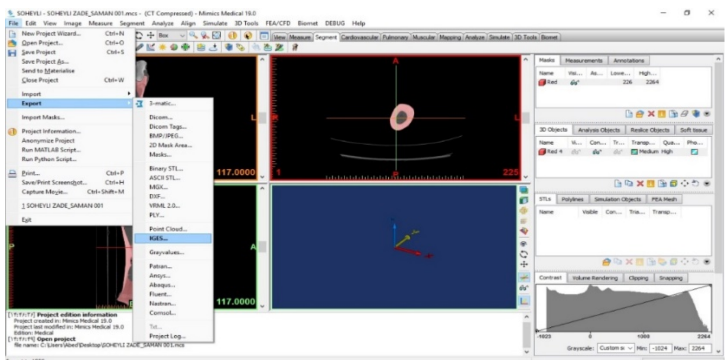


Figure 34: MIMICS output in Abaqus

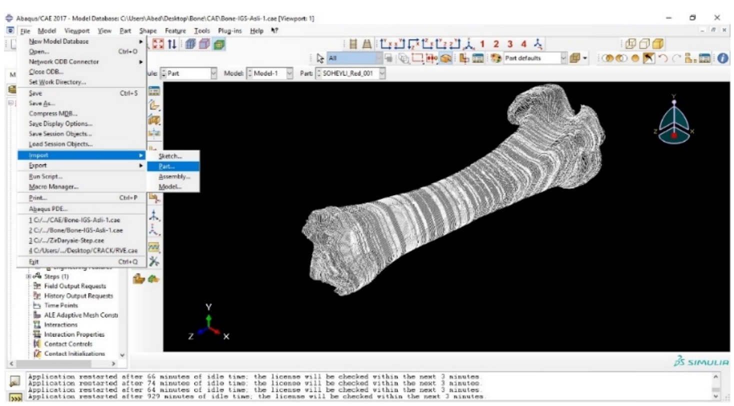


Figure 35: meshing 1

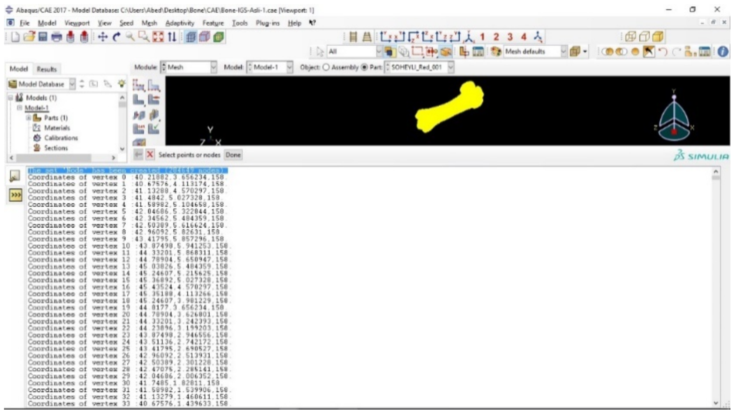


Figure 36: geometrical specification output 1

In this stage, a total of 215713 nodes with definite spatial specifications are achieved. Now, the match between the Abaqus output spots of the meshed segment and the MIMICS output spots is evaluated. After investigation, it was determined that 85% of the spots match and the closest spot is selected for the rest of the spots through writing a code in Matlab. Now, all the spots feature definite properties (density and elasticity module). It is worth mentioning that the density was coded and obtained based on Hounsfield value that was already defined for all of the corresponding specifications and also through using the density formula proposed by Taylor (2002)[2]; moreover, elasticity module was coded and obtained via making use of three approaches posited by Carter, Keller and Morgan (Roger Schols, Folk Hoflma, 2013; Rashia Begum, Arumaikkannu, 2015). In the next stage, every four spots were considered as an element and the mean specifications (density and elasticity module) were assigned as properties to each element via coding in Matlab.

x	y	z	HU	Density	Carter's module of elasticity
15.9961	-4.1133	-223	353	1163.792	5974001295
16.4531	-4.1133	-223	379	1175.856	6161715309
16.9101	-4.1133	-223	319	1148.016	5734334628
14.168	-4.5703	-223	234	1108.576	5163398288
14.625	-4.5703	-223	393	1182.352	6264401444
15.082	-4.5703	-223	443	1205.552	6640443006
15.539	-4.5703	-223	494	1229.216	7039208413

[2] Mimics Student Edition Course Book(Mimics SE) from WWW.MATERIALISE.COM

```
*
** Section: Section-431457
*Shell Section, elset=Set-431457, material=Material-431457
,
** Section: Section-431458
*Shell Section, elset=Set-431458, material=Material-431458
,
*END PART
**
** MATERIAL PROPERTIES
**
**
*Material, name=Material-1
*Density
1186.528,
*Elastic
6353414566.49592, 0.33
*Material, name=Material-2
*Density
1177.8151111111,
*Elastic
6224489288.27277, 0.33
*Material, name=Material-3
*Density
1171.88622222222,
*Elastic
6128757657.89271, 0.33
*Material, name=Material-4
*Density
1319.48977777778,
*Elastic
8796225378.71345, 0.33
*Material, name=Material-5
*Density
1438.32533333333,
*Elastic
11314320830.0002, 0.33
```

In tables (10) to (12), a summary of the obtained results has been given in the form of natural frequencies obtained respectively through approaches put forth by Carter, Keller and Morgan.

Table 10: natural frequencies based on Carter's approach

Frequency	F1	F2	F3	F4	F5
No.	670	845	1112	1618	1895

Table 11: natural frequencies based on Keller's approach

Frequency	F1	F2	F3	F4	F5
No.	660	830	1120	1710	1990

Table 12: natural frequencies based on Morgan's approach

Frequency	F1	F2	F3	F4	F5
No.	668	850	1220	2322	2530

Laboratory Results:

Next, laboratory modal analysis was used to determine the inherent dynamic characteristics of bone (natural frequencies).

Modal analysis can be done in two ways, namely experimental and theoretical. The finite element method was previously applied to obtain the natural frequency range. Modal test operation includes the measurement of frequency response function or responses of a structure subject to impact.

A fixed spot is taken into account for measuring the system response in hammer test so that an exact model of the system behavior could be obtained following which the stimulation spot is changed every time. In summary, the laboratory modal analysis is composed of three stages, named test preparation stage, response frequency measurement and identification of modal parameters. The preparation includes the selection of the structure support conditions, type of the stimulation, software used for the force measurement, identification of error resources and so forth.

- **Hammer Test:**

The natural frequency of a real cow tibia was examined using hammer test under free-free boundary conditions. It is worth mentioning that the specimen was suspended in the air using soft straps so that the free-free boundary conditions could hold.

To obtain the natural frequencies of a bone sample, the instructions should be followed as listed in table (13) regarding stimulation spot, installation of accelerometer and impact direction, in separate.

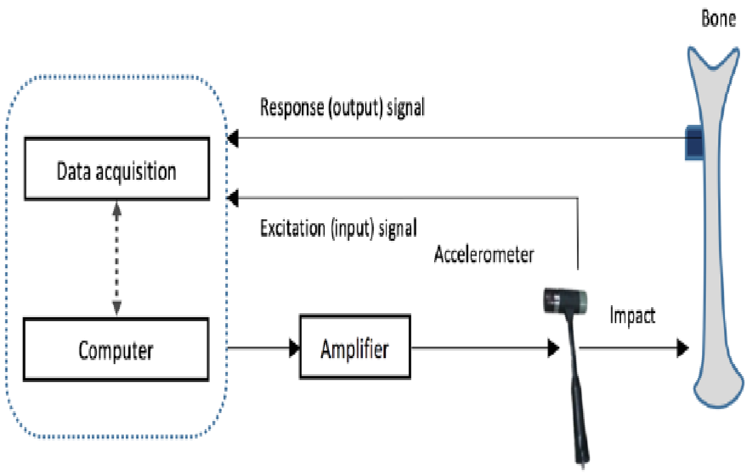


Figure 36: hammer test

Table 13: stimulation spot, accelerometer installation and impact direction

Mode type	Stimulation spot (impact)	Accelerometer installation spot	Impact direction
Mode One	Stimulation in proximal region	Accelerometer in the medial position	Vertical impact
			Horizontal impact
			Longitudinal impact
Mode Two	Stimulation in distal region		Vertical impact
			Horizontal impact
			Longitudinal impact
Mode Three	Stimulation in proximal region	Accelerometer in the distal position	Vertical impact
			Horizontal impact
			Longitudinal impact
Mode Four	Stimulation in distal region		Vertical impact
			Horizontal impact
			Longitudinal impact
Mode Five	Stimulation in proximal region	Accelerometer in the proximal position	Vertical impact
			Horizontal impact
			Longitudinal impact
Mode Six	Stimulation in distal region		Vertical impact
			Horizontal impact
			Longitudinal impact

Table (14) presents the laboratory results in separate.

Table 14: laboratory results

Sample positions	Impact direction	Impact point-accelerate metre point						Frequency 1	acceleration	Frequency 2	acceleration	Frequency 3	acceleration	Frequency 4	acceleration	Frequency 5	acceleration
		Proximal-medial	Distal-medial	Proximal-distal	Distal-distal	Proximal-proximal	Distal-proximal										
Case 1: Caudal-cranial	Vertical	1	4	7	10	13	16	662	0.00504	832	0.5412						
	Horizontal	2	5	8	11	14	17	674	0.01178	830	0.009982	1112	0.0007339				
	Longitude	3	6	9	12	15	18					1126	0.0002075			1992	0.00209
Case 2: Medial-lateral	Vertical	19	22	25	28	31	34	658	0.8009								
	Horizontal	20	23	26	29	32	35	658	0.02811	832	0.0007417						
	Longitude	21	24	27	30	33	36	670	0.005143	844	0.000198			1712	0.001009		
		Mean value						664.4		834.5		1119		1712		1992	

Data Analysis:

One of the most important constraints of the similar studies is the number of the limited materials constituting the bone and the discontinuous distribution of them. The present study endeavored to obtain a continuous distribution via specifying materials for every individual bone spot. There were used 250 thousand spots for which spatial information (x, y, and z) and Hounsfield value (indicating the density in each part) and density and elasticity module were determined so that a model could be obtained that is continuous in terms of material specification.

After specification of materials (using Matlab and Abaqus links) and determination of each element, the natural frequencies were obtained under free-free conditions. In the next stage, the approaches adopted by various individuals for obtaining natural frequencies were investigated so as to figure out which approach gives the results closer to the laboratory estimations. As it is observed in table (15), the shift of approach from discrete to continuous causes the results to get closer to the laboratory estimations and this is indicative of the appropriateness of the continuous approach (material assignment individually to each bone spot). In the article by Roger Schols et al (2013) who investigated and validated the density-elasticity relations for the finite element modeling of the human pelvis using modal test, a mean Young Modulus was obtained for each element through relating the specifications of each element to the CT-Scan data as well as by defining three Young modulus relations for density that were respectively presented by Carter, Keller and Morgan. It has to be pointed out that they had used Taylor's article to calculate density variations (as was done herein) and the modal analysis was carried out using Ansys Software so that the natural frequencies and the bone shape modes could be determined. Then, it was concluded that the elasticity-density relation offered by Morgan can better demonstrate the match between the natural frequency obtained by finite element model and the natural frequency obtained using modal test of the given sample. In this study, as well, the results of the investigation of the three approaches indicated that the result differences are smaller in Morgan's approach than the other two approaches. The analysis was executed in Abaqus Software.

Table 15: the obtained results

Row	Isotropic/Orthotropic	Kind of Mapping	Material	Software	F1	F2	F3	F4	F5	Note	Difference
1	heterogeneous isotropic elastic	One Material according to Previous Article	E=18.7 Gpa, P=1670 Kg/m3	ABAQUS Software	692	868	1190	1843	2007		-32
2		With 42 Different Material Group base on	HU between -1018 to 1970	ABAQUS Software	680	851	1150	1715	1987	Carter&Hayes	-20
		Continues Distribution	9000 Point	ABAQUS Software	670	845	1112	1618	1895		-10
					681	856	1210	1952	2224	Keller	-21
					668	850	1220	1856	2150	Morgan	-8
3	Experimental Result	Result			660	830	1120	1710	1990	Experimental	0

Conclusion

As it was mentioned, the continuous distribution surely gives results more acceptable than the prior approaches but the laboratory results can be applied to optimize the formula pertaining to density and elasticity module so that more favorable results could be obtained.

In the relations pertinent to Taylor's formula of density, a coefficient, assumed to be equal to 0.4641, can be optimized so that the density values can get closer to the laboratory findings.

$$\rho_{eff} = AHU + 1000(\frac{kg}{m^3}) \quad \text{(Relation 1)}$$

It is noteworthy that in the article presented by Taylor in 2002, A coefficient has been set equal to 0.523 that was reduced to 0.464 in the later articles after performing investigations and obtaining laboratory results and optimization of the corresponding values. It was set equal to 0.464, as well, in the present study. Thus, the intended value can be repeatedly changed through coding in Matlab so that results can be obtained that are closer to the laboratory findings. Interestingly, the subject has not been much debated in the articles printed previously. So, it can be investigated as a novel topic in an investigation of the density and Hounsfield coefficient relations.

It has to be pointed out that the aforementioned topic holds for the elasticity module, as well. In the aforesaid relation, values pertaining to a and b coefficients can be assumed in such a way that values could be obtained through exercising some optimization that are closest to the laboratory results.

$E = a\,\rho^b$ (Relation 2)

Genetic algorithm can be used to deal with the issue so that a and b values could be obtained for the studied sample that would be of a considerable contribution to the investigation of the mechanical properties of bone.

Reference

1. A. Gupta, K. M. Tse, Vibration Analysis of Femur Bone Using Elmer, Journal of Engineering Science, Special Issue on ICMTEA 2013 Conference, December (2014) 12 – 19
2. Abaqus Software, Help, 2017.
3. Amaziah Walter Otunyo, Umenwo Lucky E. and Sam Kingsley, Machine Crushed Cow Bones as a Partial Replacement of Fine Aggregates in Light Weight Concrete, Journal of Engineering and Applied Science, 2014.
4. Arild Aamodet, Kjell, Norway, Determination of the Hounsfield value for CT-based design of custom femoral stems, 2010.
5. B. Helgason, E. Perilli, E. Schielo, F. Taddei, S. Brynjolfsson and Viceconti M. Mathematical relationship between bone density and mechanical properties: a literature review. Clinical Biomechanics, (23);135 46, Feb.2008.
6. Bekir Bediz, H. Nezat, Vibration measurements predict the mechanical properties of human tibia. Elsevier Publication ,2010
7. Biomechanics of hard tissues, Modeling, Testing and Material. Edited by Andreas Wager .2010.
8. Chen, Schule, A new approach for assigning bone material properties from CT images into finite element method, Elsevier Publication, 2010.
9. Christian Wong, Peter Mikkelsen, Leif Berner Hansen, Tron Darvann, Peter Gebuhr, Finite element analysis of tibial fractures, MAY 2010.
10. Department of Mechanical Engineering, b Department of Manufacturing Engineering, College of Engineering Guindy, Anna University, Chennai, India.2009
11. Mimics Student Edition Course Book(Mimics SE) from WWW.MATERIALISE.COM.
12. P. Schrock A, B, ⇑, M. Lüpke a, H. Seifert a, L. Borchers C, C. Staszyk, Finite element analysis of equine incisor teeth. Part 1: Determination of the material parameters of the periodontal ligament, ELSEVIER Publication ,2013.
13. R. Neugebauer a, M. Werner A, N, C. Voigt b, H. Steinke c, R. Schols b, S. Scherer a, M. Quickert, Experimental modal analysis on fresh-frozen human hemi pelvic bones employing a 3D laser vibrometer for the purpose of modal parameter identification, Elsevier Publication, 2011.
14. Rashia Begum S. A., Arumaikkannu Gb., Finite Element Analysis in Additive Manufactured Customized Bone Scaffold, 2015.
15. Roger Schols, Folk Hoflma, Validation of density–elasticity relationships for finite element modeling of human pelvic bone by modal analysis, Elsevier Publication, 2013.
16. S. N. Khan, R. M. Warkhedekar, Analysis of Hounsfield Unit of Human Bones for Strength valuation. Elsevier Publication, 2010
17. Suk-Hun Kim, Seung-Hwan Chang, Ho-Joong Jung, the finite element analysis of a fractured tibia applied by composite bone plates considering contact conditions and time-varying properties of curing tissues, ELSEVIER Publication, 2010.
18. Taddi, Enrico, the material mapping strategy influences the accuracy of CT-based finite element models of bones: An evaluation against experimental measurements, ELSEVIER Publication, 2007.
19. Taylor, Roland, Ploeg, Determination of orthotropic bone elastic constants using FEA and model analysis. Elsevier Publication, 2002.