CHAPTER 3

RESEARCH MRTHODOLOGY

3.1 Software and 3D model reconstruction

3.1.1 Computer software

- Simpleware ScanIP. (3D Image Visualization and Processing Software)
- Dassault Systemes SolidWorks 2015 Corporation. (3D models reconstruction)
- Dassault Systemes Abaqus CAE 6.13 (solving finite element analysis and sophisticated engineering problems)
- Microsoft office Excel 2016 (making and compilation results)
- IBM SPSS 17.0 statistics (statistical analysis)

This study used simulated device from Lenovo company (Processor: Intel(R) Xeon(R) CPU E5-2640 v3 @ 2.60GHz 2.60 GHz, installed memory (Ram): 64.0 GB, system type: 64-bit Operating system) (Fig. 3.1).

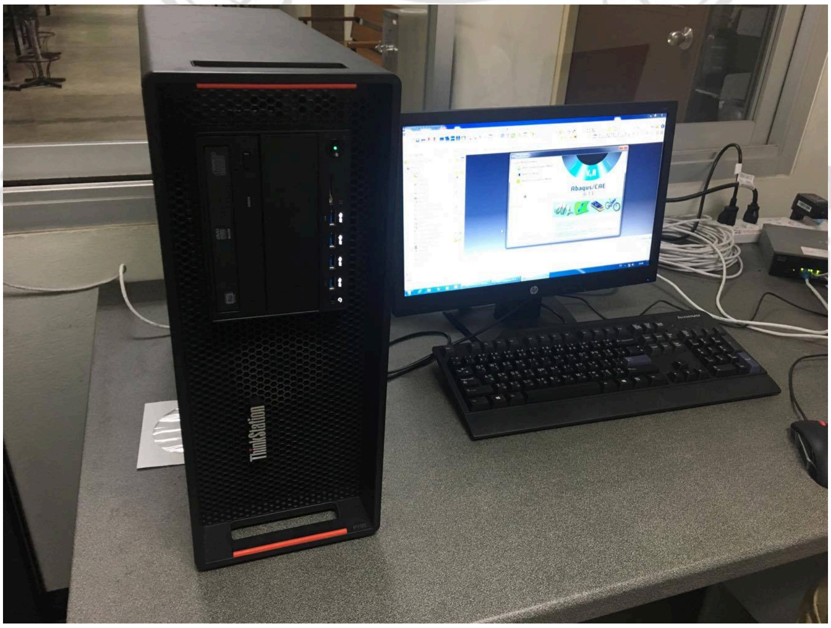

Figure 3.1 simulation device

3.1.2 3D model reconstruction

This research used complete denture from the edentulous patient and supported by mini dental implant.

3.1.3 Human edentulous mandibular

Human edentulous mandibular used a CT scans (Dicom image files), the edentulous patient from faculty of dentistry Chiang Mai university. This dicom image files can identify only the outer contour and surface of the human edentulous mandibular (cortical bone) (Simpleware ScanIP). Cancellous bone reconstructed simply by used round 2 millimeters constant thickness from the outside layer along the edentulous mandibular arch (SolidWorks 2015, DassaultSystemes SolidWorksCorp. It shown in (Fig. 3.2-3.4)

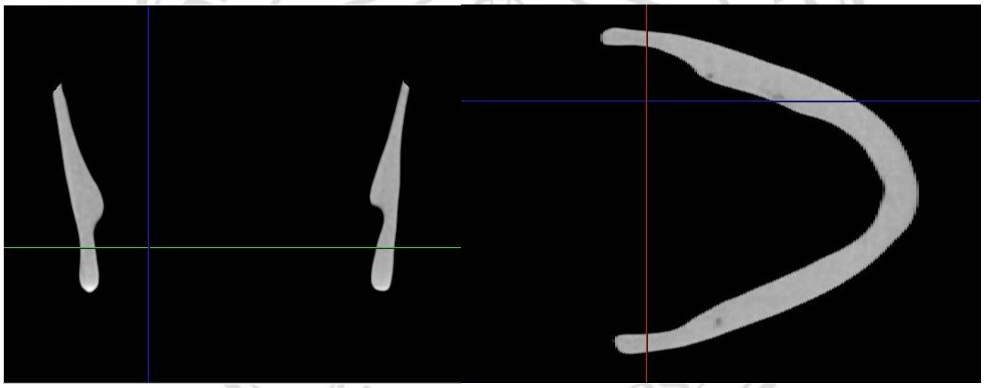

Figure 3.2 Human edentulous mandibular CT scan

Figure 3.3 Human edentulous mandibular model

Figure 3.4 Human edentulous mandibular model with cortical and cancellous bone layer

3.1.4 Complete denture

Modifying from the same CT scans of the edentulous patient. this complete denture model can't use directly because of the rough and complex outer surface. Moreover, STL file could not modify directly and easily. Consequently, complete denture was reconstruction and modification in Solidwork to make it simplify. As shown in (Fig 3.5-3.6)

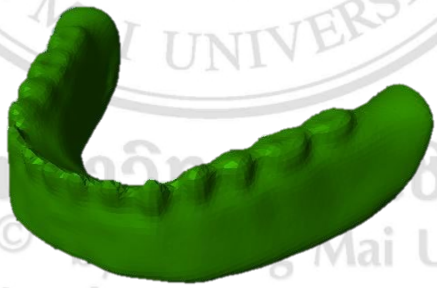

Figure 3.5 Denture model from dicom files directly (STL files)

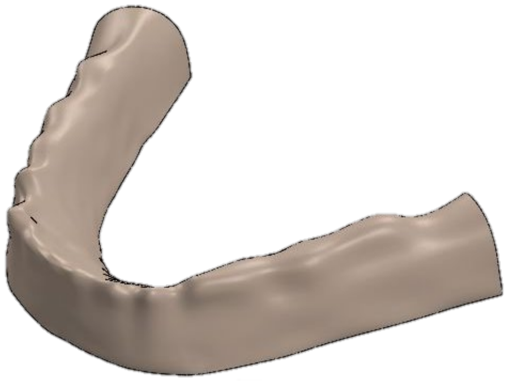

Figure 3.6 Modified denture model from STL files

3.1.5 Gingiva

Simplify gingiva were generated by using around 2-millimeter thickness along the anterior and posterior edentulous mandibular area (modified by mandibular shape) figure 3.7.

Figure 3.7 Gingiva model

3.1.6 Mini-dental implant and others component.

Mini-dental implant (MDIs) were used correlated with (PW+ MDI, PW+ Nakhon Pathom, Thailand) used 2.75-millimeter diameter (measure from the fine thread position) 12 millimeters in length (if measure from the first to the last fine thread, the length is 10 millimeters) it shown in figure 3.4.

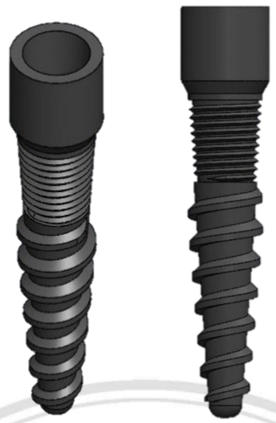

Figure 3.8 Perspective and side view of mini dental implant

3.1.7 Equator attachment (metal head of mini dental implant)

Reconstruction correlated with (Rhein83, Bologa, Italy). As shown in figure 3.9 - 3.11 with the Oring, Oring was connected between metal head and abutment part. Abutment was attached with MDI body. All models were reconstructed from the preliminary data.

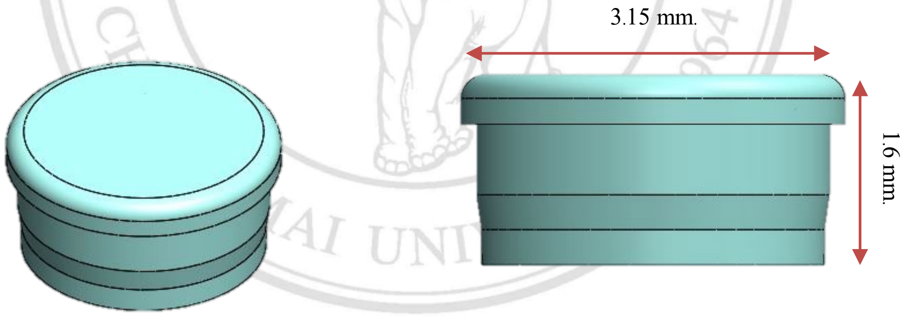

Figure 3.9 Equator attachment part (Metalhead)

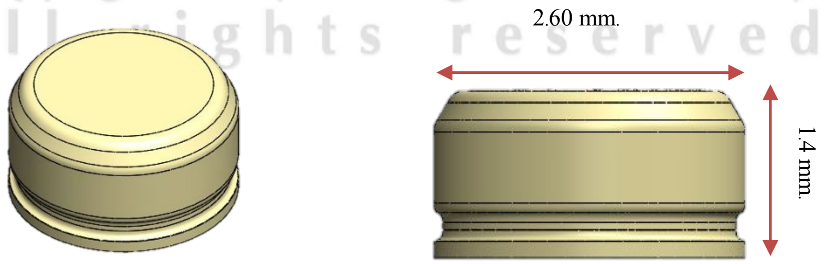

Figure 3.10 O-ring attachment part

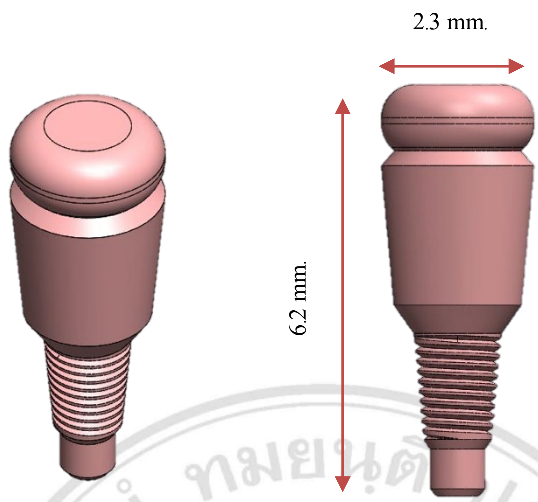

Figure 3.11 Abutment part

3.2 Methods

3.2.1 Finite element analysis procedure

3.2.1.1 Group study classification

In this research, 10 group study were divined by number of MDI and position of implantation as follow: Group1, implantation 2 MDIs in both side of edentulous mandibular at incisor area. Group2, implantation 2 MDIs in both side of edentulous mandibular at canine area. Group3, implantation 2 MDIs in both side of edentulous mandibular at premolar area. Group4, implantation 3 MDIs at midline and both side of edentulous mandibular at incisor area. Group5, implantation 3 MDIs at midline and both side of edentulous mandibular at canine area. Group6, implantation 3 MDI at midline and both side of mandibular at premolar area. Group7, implantation 4 MDIs 2 at incisor area and 2 at canine area in both side of edentulous mandibular. Group8, implantation 4 MDIs 2 at incisor area and 2 at premolar area. Group9, implantation 5 MDIs at midline, 2 at incisor area and 2 at canine in both side of edentulous mandibular. Group10 the last group, implantation 5 MDIs at midline, 2 at incisor area and 2 at premolar area in both side of edentulous mandibular. In all group MDIs, had been connected with overdenture. Table 3.1

Table 3.1 Group study classification

Group study	Number of implantation	description
Group 1	2	Implantation at incisor area
Group 2	2	Implantation at canine area
Group 3	2	Implantation at premolar area
Group 4	3	Implantation at midline, incisor area
Group 5	3	Implantation at midline, canine area
Group 6	3	Implantation at midline, premolar area
Group 7	4	Implantation at incisor area, canine area
Group 8	4	Implantation at incisor area, premolar area
Group 9	5	Implantation at midline, incisor area, canine area.
Group 10	5	Implantation at midline, incisor area, premolar area.

Figure 3.12 Edentulous mandibular with 2 mini dental implants at incisor area (group1)

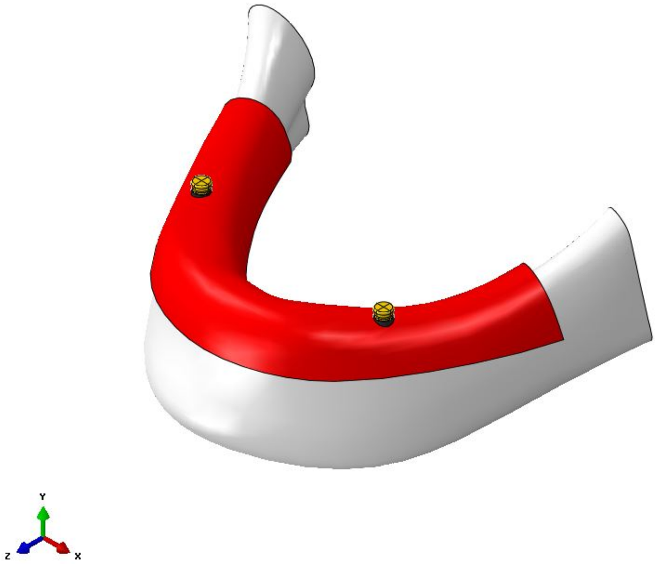

Figure 3.13 Edentulous mandibular with 2 mini dental implants at canine area (group2)

Figure 3.14 Edentulous mandibular with 2 mini dental implants at premolar area (group3)

Figure 3.15 Edentulous mandibular with 3 mini dental implants at midline and incisor area (group4)

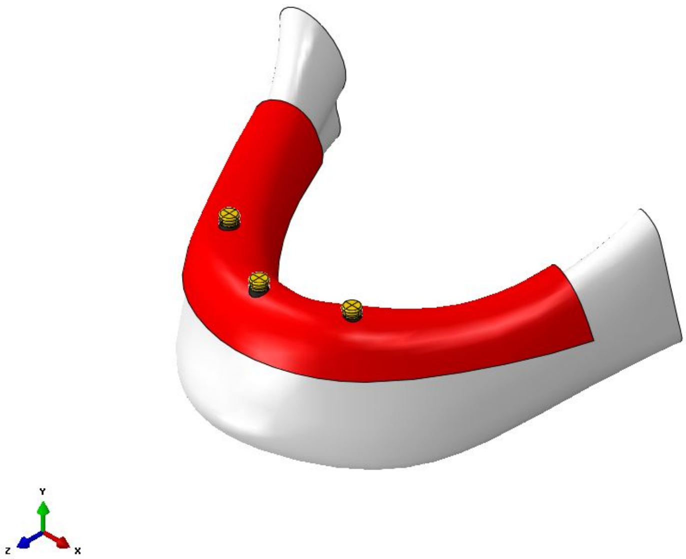

Figure 3.16 Edentulous mandibular with 3 mini dental implants at midline and canine area (group5)

Figure 3.17 Edentulous mandibular with 3 mini dental implants at midline and premolar area (group6)

Figure 3.18 Edentulous mandibular with 4 mini dental implants at incisor and canine area (group7)

Figure 3.19 Edentulous mandibular with 4 mini dental implants at incisor and premolar area (group8)

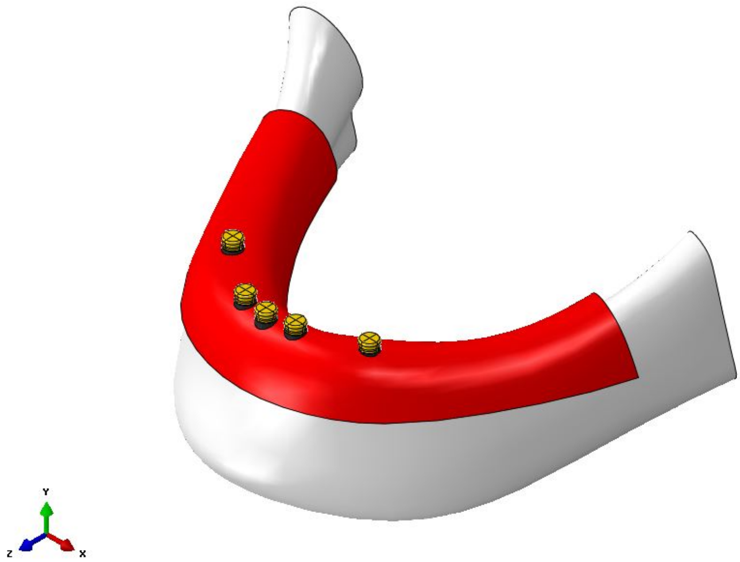

Figure 3.20 Edentulous mandibular with 5 mini dental implants at midline, incisor, and canine area (group9)

Figure 3.21 Edentulous mandibular with 5 mini dental implants at midline, incisor, and premolar area (group10)

3.2.1.2 Finite element modelling and Material properties

After FEA edentulous mandibular, MDI and other components were reconstructed relate with the preliminary data by Solidwork 2015 (Dassault Systemes Solid Works Corp.) that reported in 3D model reconstruction compendium. All of 3D models were not subtracted (cutting the excess model part) in Solidwork, all parts were only installed in the right position to make the template model, figure 3.15. The template model was exported from Solidwork in Parasolid file to AbaqusCAE 6.13 (FEA simulated, acknowledge Abaqus license from School of AMME, the university of Sydney). In AbaqusCAE, the template model was copied and divined to 10 group follow the classified group. In each group, the edentulous mandibular model was modified to make an implantation position (subtracting) according to 10 groups, figure 3.16.

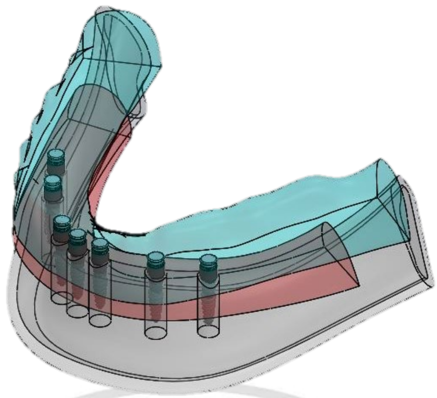

Figure 3.22 FEA template model (Solidwork)

Further, around the implantation areas have the cylinder parts 4 millimeters' diameter and height up to surface of edentulous mandibular in all groups. The cylinder parts were designed for make the peri-implant area, figure 3.17. Hence, all models were interested on the bone surrounding MDIs (cortical and cancellous bone).

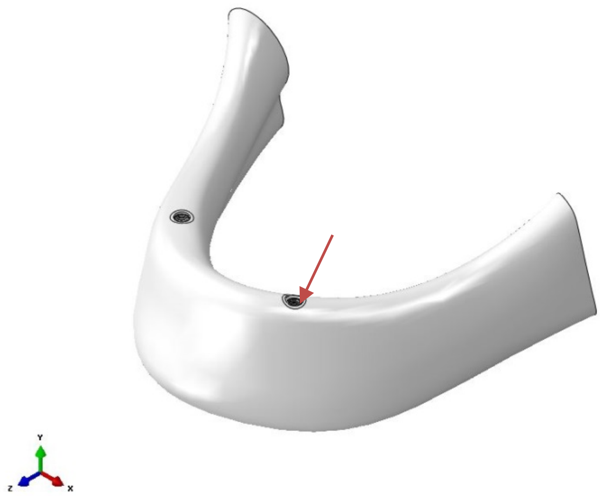

Figure 3.23 Sample model from one group showed the subtracted area on edentulous mandibular at the red arrow (dental implant)

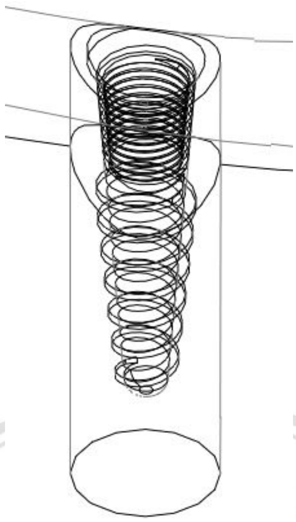

Figure 3.24 Peri-implant area in cortical and cancellous bone on edentulous mandibular

The materials and fabric were acquired from many literatures in previous studies. The material properties show the different physical and mechanical responds and influenced the stress and strain distribution in a structure. All of materials were considered as isotropic and homogeneous and that they have elastic material behavior characterized by two constant values of Young’s modulus and Poisson’s ratio. The 2 constant values were presented in table 4. So, this study can be used the linear case because before the FEA test is a small deformation.

Table 3.2 Mechanical and Material properties used in this study

Material	Model structure	Young’s Modulus (*MPa*)	Possion ratio	References
Cortical bone	Mandibular	13,700	0.3	Barbier et al. (1998)
Cancellous bone	Mandibular	1,370	0.3	Barbier et al. (1998)
Titanium (Ti-6AI-4V)	Implant body, abutment	103,400	0.35	Sertgoz and Guvener (1996)

Table 3.2 (Continued)

Material	Model structure	Young's Modulus (*MPa*)	Possion ratio	References
Rubber	O'ring	5	0.45	Chun et al. (2005)
Stainless steel	Metalhead	19,000	0.31	Barao et al. (2009)
Acrylic	Denture	8,300	0.28	Darbar et al. (1995)
Mucosa	Gingiva	680	0.45	Barao et al. (2008)

3.2.1.3 Meshing quality Finite element analysis and model validation.

After modifying all of the models in Abaqus CAE, the linear tetrahedron element type (C3D4 with 4 node) was used in every part since models have irregular shape. Moreover, in some models have many sharp areas. The models were not used the simplify element shape as a hexahedron element (C3D8R, 8 node), figure 3.18.

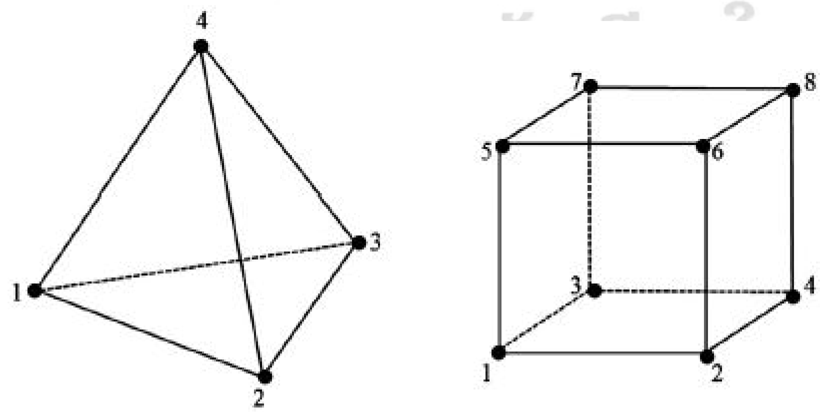

Figure 3.25 Linear tetrahedron element type and hexahedron element type, respectively

The stress and strain distribution surround the peri-implants were interested. Hence, the number of these parts should fine enough by convergence test (Fig 3.19) to reach accurate results. In convergence test, only mandibular part was tested. The meshing quality for validation the model was tested many time and refined in the focusing area until the results did not change. The elements and nodes in each component were presented in table 5. the number of element and node in each group were not equal because models were meshed by a geometry of model. In some parts were modified (partition) before meshing for made a good quality of elements, (Fig 3.20).

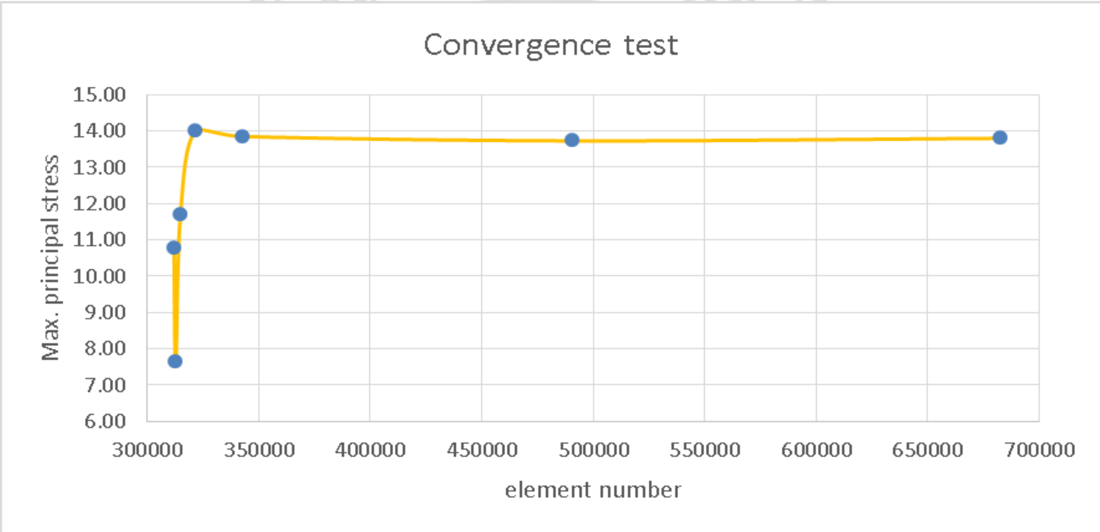

Figure 3.26 Convergence test element

Table 3.3 Number of elements in example study groups in convergence test

	Mandibular element	Max principal stress (*MPa*)
1	311926	10.78
2	312680	7.65
3	314750	11.69
4	321456	14.00
5	342925	13.85
6	490531	13.73
7	682926	13.80

Table 3.4 Number of elements and nodes in each component of the models

Model components	Number of element (C3D4)	Number of node
Group1	2,258,172	405,549
Group2	2,245,995	402,772
Group3	2,272,021	408,331
Group4	2,259,310	407,590
Group5	2,263,346	407,921
Group6	2,297,658	416,031
Group7	2,341,062	424,901
Group8	2,311,656	420,362
Group9	2,303,569	419,180
Group10	2,320,143	420,432
Denture	563,639	109,128
Gingiva	342,799	65,042
Abutment	41,437	8,128
Implant body	101,681	20,890
Metal head	24,279	5,811
O ring	27,395	6,438

*Edentulous mandibular in each group, not equal because the number of implantation (cortical and cancellous bone were included)

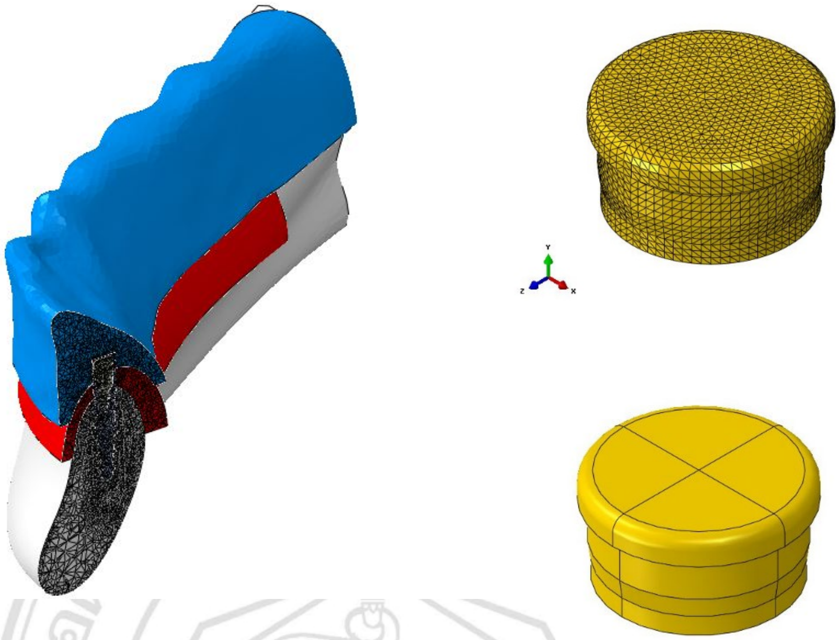

Figure 3.27 Meshing and partitioning model to good quality elements

3.2.1.4 Interface Conditions.

The model interfaces were applied in 2 types. First, between mandibular bone and MDI was set as a surface to surface contact and used tangential behavior (friction condition) used friction coefficient 0.3 [62]. The contact between the components in MDI structure was set as same as the contact of mandibular bone and MDI but used the different friction coefficient 0.5 [63]. Furthermore, the others contact in this study, used constraint (ties contact), to fix other components (without relative move) for simplify study [15].

3.2.1.5 Boundary Conditions.

This study, boundary conditions were set as fixed in edentulous mandibular like previous studies [15]. Edentulous mandibular model was constrained fix in all direction (not move and rotate) at posterior area in both of cortical and cancellous bone. It was showed in figure 3.20

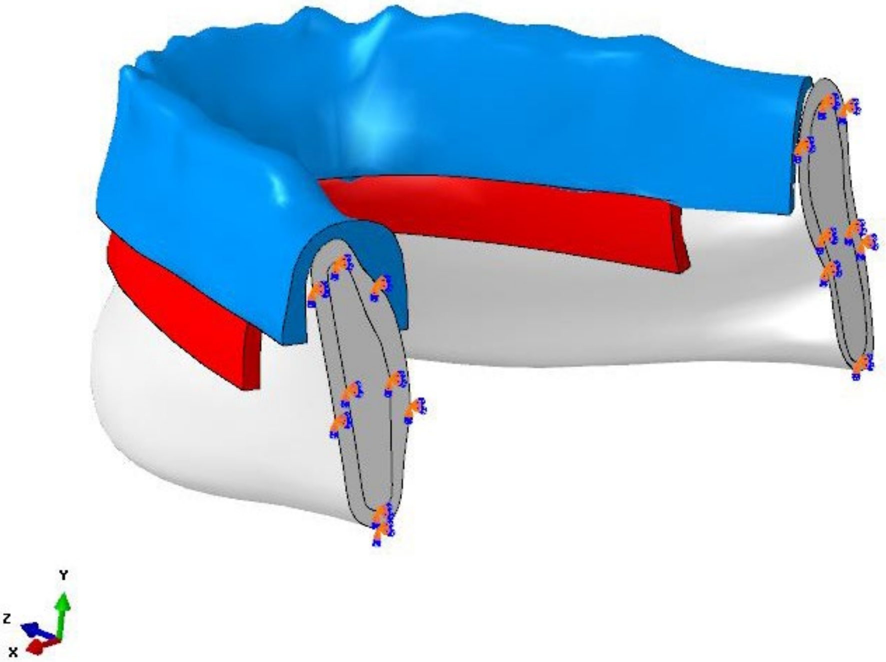

Figure 3.28 Boundary conditions were fixed at posterior area

3.2.1.6 Loading Conditions.

In this study, loading 200 N (Newton) in axial (static loading) was applied to the overdenture part. In the previous study, the loading was the normal length of occlusal force mastication and nearly reach the maximum of loading condition for implant overdenture patients. The load was distinguished in 2 cases: bilateral loading and unilateral loadings (Figure 3.21).

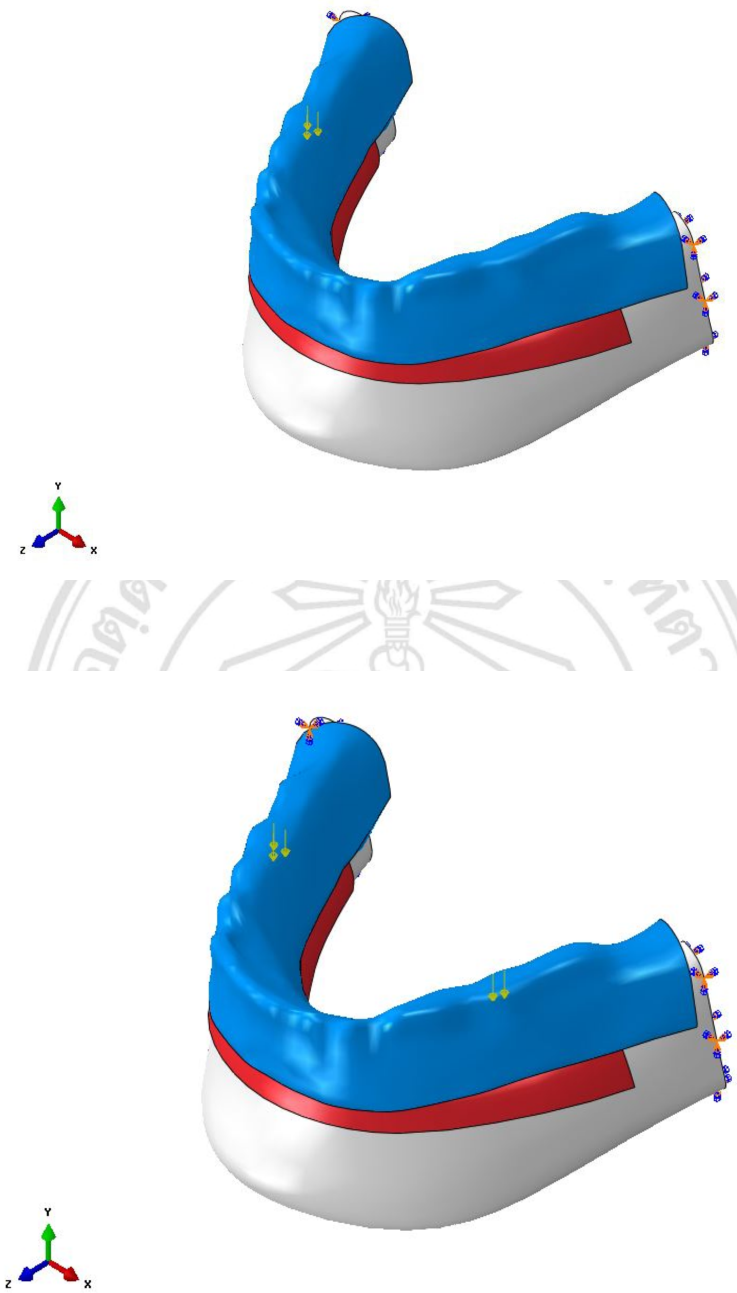

Figure 3.29 Unilateral load condition and Bilateral load condition, respectively

Unilateral load condition was applied directly on the right side of edentulous mandibular on the first molar area with distribution load (used concentration force with multiple nodes). The left side of edentulous mandibular was considered non-loading side to distinguish the stress and strain distribution with bilateral load condition.

Bilateral load condition was applied directly on overdenture component with distribution load (used concentration force with multiple nodes) on the first molar area both side of edentulous mandibular. This idea was modified from Elsyad et al. study used a metal bar to apply load that the procedure has been modified from [64]

3.3 Data and statistical analysis.

The finite element edentulous mandibular model (peri-implant area) and other components were made by using finite element program (AbaqusCAE 6.13). this program not only simulation but was used to observe and measure the mechanical parameters like: Von Mises stress, Maximum principal strain, pressure, displacement, etc. in this study, focusing on Von Mises stress and Maximum principal strain around peri-implant area. The visual display of this program was presented by changing the value to show in color. The red and orange color are showed the high values and the blue or green are showed the low values, on Mises stress equation and principal strain are normally used in finite element analysis program. The Von Mises stress equation is showed follow by:

$$\sigma_v = \sqrt{\frac{(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{xy}^2 + \tau_{zx}^2)}{2}} \quad (1)$$

The x, y and z indicate the direction of the stress. σ_x, σ_y, and σ_z indicated the individual stress in x, y, and z direction. τ_{xy}, τ_{yz} and τ_{xz} indicated the shear stress. The principal strain equation was obtained as follows:

$$\varepsilon_{1,2} = \frac{\varepsilon_x + \varepsilon_y}{2} \pm \sqrt{\left(\frac{\varepsilon_x - \varepsilon_y}{2}\right)^2 + \left(\frac{\gamma_{xy}}{2}\right)^2} \quad (2)$$

The x, y and z indicate the direction of the strain. ε_x and ε_y indicated the individual strain in x and y direction. γ_{xy} indicated the shear strain. These equations were used in AbaqusCAE to calculate the results.

Hence, after simulation there are many approaches to present Von Mises stress and principal strain of peri-implant area. The volume-averaged technique was adapted to find the peri-implant area. This technique was used in the previous study about the bone mass density at the bony tissue quality through the remodeling process in specific area [65]. This technique measurement is calculated by:

$$\bar{\rho} = \frac{1}{V}\int_V \rho dV \approx \frac{1}{\sum_{e=1}^{n} V_e}\sum_{e=1}^{n} \rho_e V_e \qquad (3)$$

That V_e is the volume of element e, n is the total number of elements in a selected area of interest and ρ_e is the Von Mises stress or principal strain of element e.

Furthermore, this study used a geometric mean technique to analysis volume averaged Von Mises stress and principal strain because of the various results. Geometric mean is used in many fields, mostly in financial reports or a fluctuating of investments. it is the geometric mean that gives the average rate [66],[67],[68],[69]. So, this idea was used to manage all of various result. The geometric mean function normally has in Microsoft Excel program function that it can be used by type: =GEOMEAN in the function space. Finally, all data were analyzed by using statistic analytical software (SPSS version 17.0, SPSS Inc., Chicago, IL, USA) by using one-way ANOVA. If significance was noted, multiple comparison analysis, the post hoc test (Tukey). A P-value < 0.05 was considered to statistically significant. Furthermore, the geometric mean was obtained as following:

$$Geometric\ mean = G_x = \sqrt[n]{\prod_{i=1}^{n} X_i} = \left(\prod_{i=1}^{n} X_i\right)^{\frac{1}{n}} \qquad (4)$$

And taking logarithm

$$\log G = \frac{1}{n}\sum_{i=1}^{n}\log X_i \qquad (5)$$

This is the mean of the logarithm of the random variable X, i.e,

$$\log G = E(\log X) = E(\log x(F)) \qquad (6)$$

Therefore:

$$G = e^{E(\log X)} = e^{[E(\log x(F))]} \qquad (7)$$