

Chapter 3

Finite Element Modeling

3.1. Methodology of Finite Element Modeling

Modeling started by create geometry model of rough surface and rigid plat. Model of geometry refer to author anylisis regarding topography of real surface. Material defined according to reference from experimental setup [1].

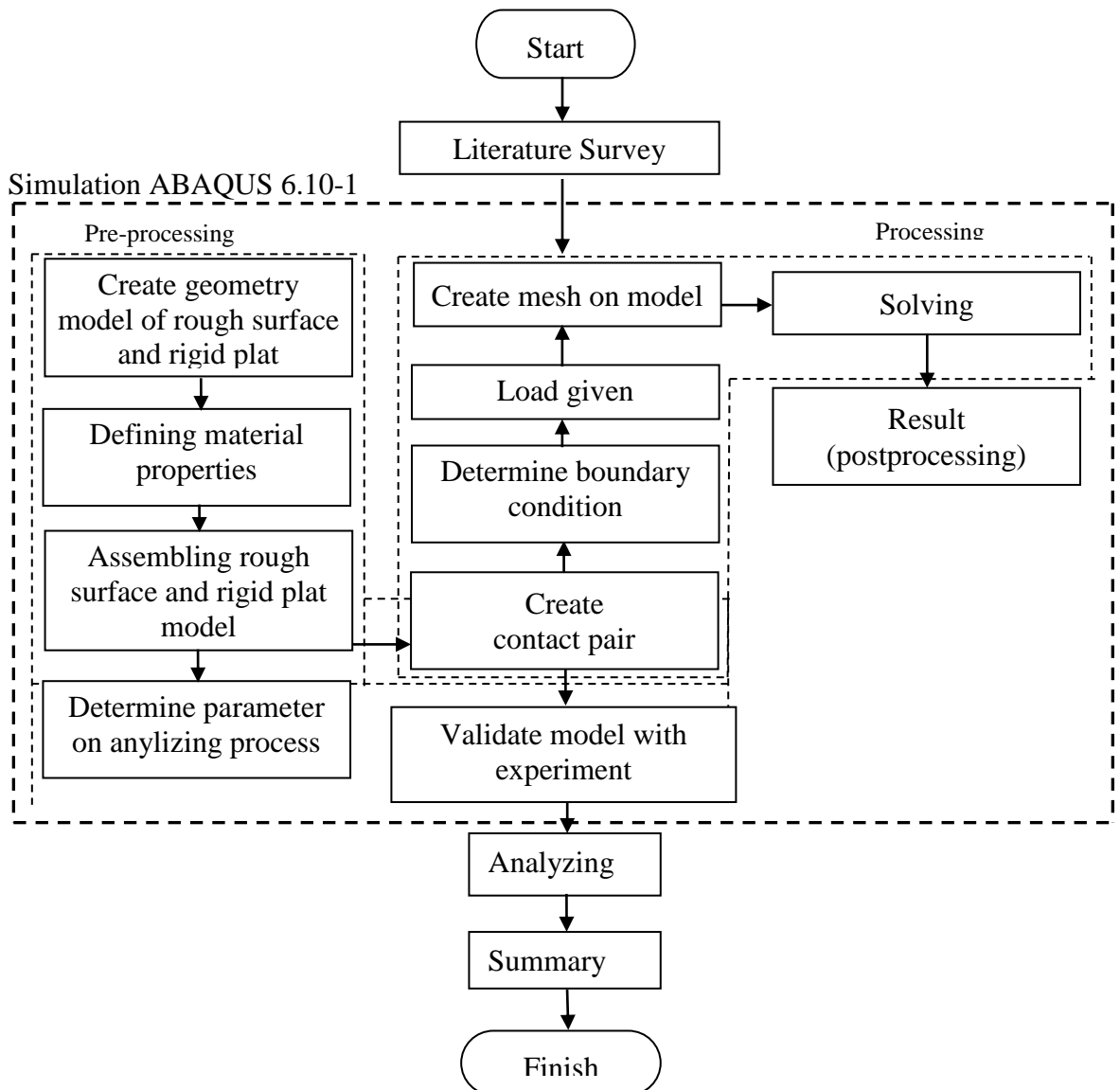


Figure 3.1: Flow chart modeling on FEM ABAQUS 6.10-1.

Determining control type and control solution are really influence on simulation result. Cautious action should be necessary in order to make the result similar with theory, especilly on determining contact pair (Fig. 3.1).

3.2 Basic Theory of Finite Element Method

The development of the computer world has been so quickly affect the areas of research and industry, so the dream of the experts in developing science and industry has become a reality. Now days, the design and analysis methods have been widely used mathematic complex calculations in many works. Finite element method (FEM) contributed many inventions in the field of manufacture and industry research, this is due to its role as a research tool in numerical simulation.

Finite element method (FEM) is a method of analysis calculations based on the idea of building a very complex object into several simple parts (blocks), or by dividing a very complex object into smaller pieces.

1. The basic concept of FEM analysis.
 - a. Making the discrete elements to obtain the deviations and forces in smaller scale from the parts of a structure.
 - b. Using elements of the continuum approach to obtain solutions from the problems of heat transfer, fluid mechanics and solid mechanics.
2. Structure analysis procedures.
 - a. Dividing the structure into pieces (elements with nodal).
 - b. Providing the physical properties of each element.
 - c. Connect the elements at each node to form an approximate system of equations for the structure.
 - d. Solving systems of equations are coupled with an unknown number of nodes (e.g., displacement).
 - e. Calculate the desired amount (e.g., strains and stresses).
3. Implementations on a computer.
 - a. Preprocessing (making the FE model, loads and constraints).
 - b. FEA solver (assembling and finishing system of equations).

- c. Post processing (show results).
- 4. Types of elements in the finite element method
 - a. One-dimensional elements (lines)

These elements include the type of spring (spring), truss, beam, pipe, etc., as shown in Figure 3.2.



Figure 3.2: Line element.

- b. Two-dimensional elements (fields)

These elements include the type of membrane, plate, shell and so on as shown in Figure 3.3.

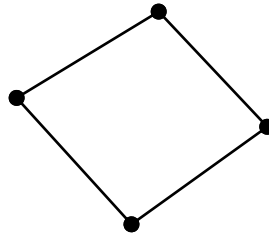


Figure 3.3: Field element.

- c. Elements of three-dimensional (volume)

These include the element type (3-D Fields-temperature, displacement, stress, flow velocity), as shown in Figure 3.4.

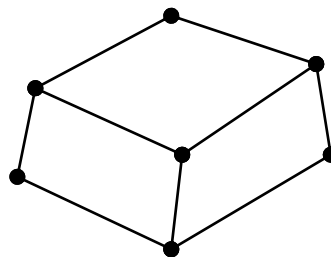


Figure 3.4: Volume element.

This element is used in two-dimensional problems that require high accuracy in analyzing the model (Fig 3.5).

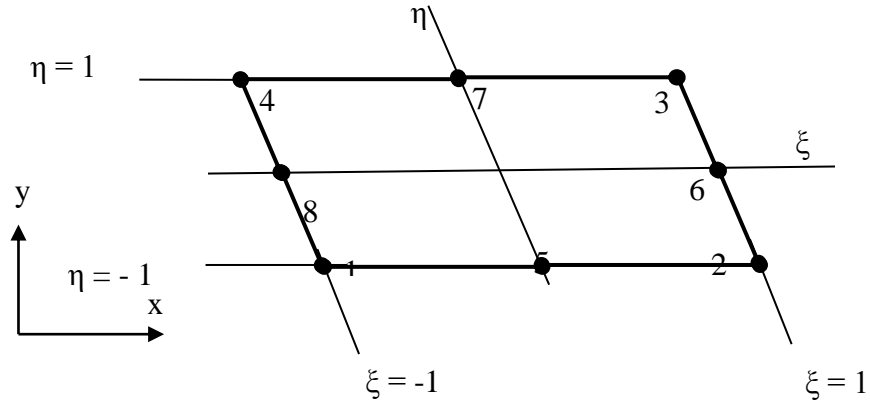


Figure 3.5: Element of quadratic quadrilateral.

There are eight points for this element. 4 points located at the corner point and 4 point in the middle. In natural coordinate system (ξ, η) , those eight function in form of (N) are,

$$N_1 = \frac{1}{4}(1 - \xi)(\eta - 1)(\xi + \eta + 1)$$

$$N_2 = \frac{1}{4}(1 + \xi)(\eta - 1)(\xi - \eta + 1)$$

$$N_3 = \frac{1}{4}(1 + \xi)(\eta + 1)(\xi + \eta - 1)$$

$$N_4 = \frac{1}{4}(1 - \xi)(\eta + 1)(\xi - \eta + 1)$$

$$N_5 = \frac{1}{2}(1 - \eta)(1 - \xi^2)$$

$$N_6 = \frac{1}{2}(1 + \xi)(1 - \eta^2)$$

$$N_7 = \frac{1}{2}(1 + \eta)(1 - \xi^2)$$

$$N_8 = \frac{1}{2}(1 - \xi)(1 - \eta^2) \quad (3.1)$$

Furthermore,

$$\sum_{i=1}^8 N_i = 1 \quad (3.2)$$

Displacement equation (u, v) is

$$u = \sum_{i=1}^8 N_i u_i, \quad v = \sum_{i=1}^8 N_i v_i \quad (3.3)$$

3.3 Specification and Geometry Problem

Authors verify model of surface from present method through applied a particular case of contact elastic deformable rough surface against hard smooth ball (Fig. 3.6.a). In ABAQUS, rough surface model that was imported from SolidWorks is portrayed in Figure 3.6.b the surface for simulation has waviness with multiple heights of asperities. Moreover, surface can easily be meshed with smaller mesh on the top of surface. This particular contact problem will be applied to evaluate the model.

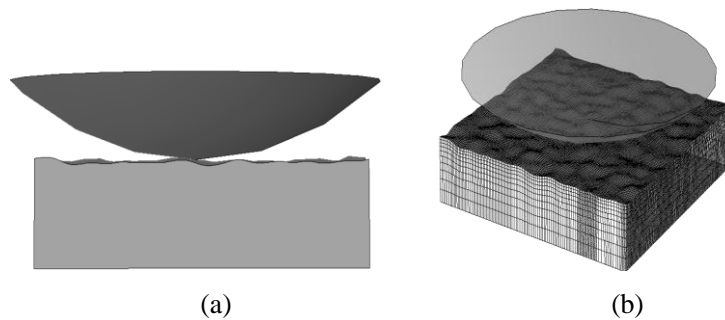


Figure 3.6: (a) Contact model of rough surface vs hard ball (b) surface geometry in ABAQUS after being imported from SolidWorks.

There will be three model used in simulation, first is sinusoidal surface, second is random rough surface, and third is experimental model transferred in finite element model. Those models are applied under static contact condition. The software package ABAQUS was used to analyze the three models created in this study.

Contact static simulation performed to determine the deformation that occurs on rough surfaces. The different types of contact interactions lead to different deformation occurs. The number of initial simulations are performed only once that aims to

determine the deformation of various types of contact interactions and compared with each other.

Rough surfaces materials used for all simulations in this study is aluminum. Properties of the material used for experimental data is elastic perfectly plastic with $E = 75.2\text{GPa}$, $\nu = 0.3$ and $\sigma_y = 85.714\text{ Mpa}$ [1] with corresponding material model is frictionless and hard contact. Result from simulation is captured to identify deformed asperity. As cross-sectional pieces surface in which some asperity is deformed as shown in Figure 3.7.

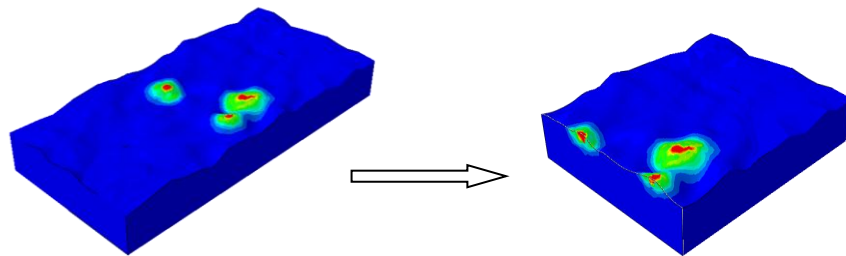


Figure 3.7: Captured surface on cross sectional area.

Modeling using the finite element method which cut surface into several elements (pieces of the surface), then reconnects elements at “nodes” as if nodes were pins or drops of glue that hold elements together. This process results in a set of simultaneous algebraic equations. Each nodes has values which represent condition of surface. Figure 3.8. show the method are used to take values from each nodes. Retrieval results are then plotted in the graph to be displayed in the next chapter.

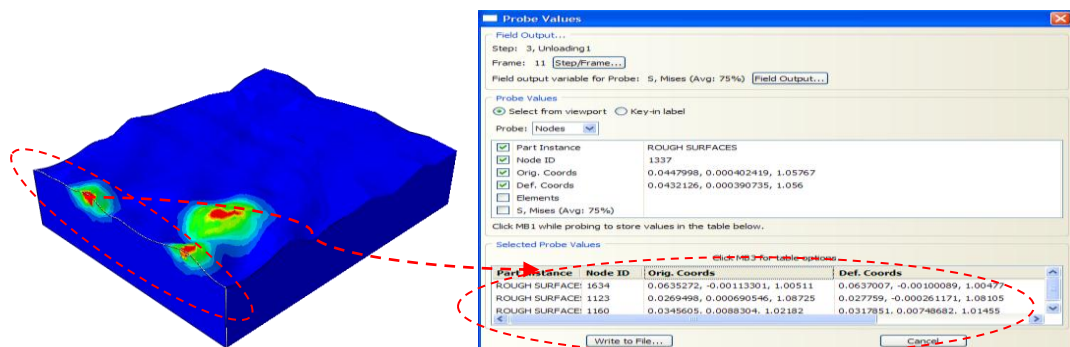


Figure 3.8: Procedures of measuring von Mises stress from each nodal.

Mesh densities were increased around the top surface area. The contact load per unit length of the contact corresponding to a given interference was retrieved by summing the reaction forces at each of the nodes in contact with the rigid plane. The contact dimension was obtained from output data files created by the software, which indicated the state of contact at each surface node. Contact pressures and internal stress values were obtained from the standard field outputs of the software. In the finite element simulations described here the term “interference” represents the approach of the upper edge of the model towards the contact plane, and zero interference corresponds to initial contact of the undeformed model at zero load. The size of each interference increment was governed by the need to maintain numerical convergence of the iterative solution process embodied in the software to solve the non-linear contact.

3.4 Description of the Modeling Method

3.4.1 Process Pre-processing from Matlab-SolidWorks

3.4.1.1 Determining Surface Geometry on Matlab

In the present study, authors work on real surface which generated in plot graphic (Fig. 3.9). Number of surface point along square side (n), length of surface along square side (r_l), root means square height (h), and correlation length in x and y coordinates cl , is determined to generate the surface. Different color on graphic plot indicate different height of surface, brighter area represent area with higher asperities while darker area represent area with lower asperities. The model is then generated in square form and according to script below:

```
N=500;
rL=2;
h=0.5;
clx=0.2;
x = linspace(-rL, rL, N);
y = linspace(-rL, rL, N);
[X, Y] = meshgrid(x, y);
```

```

Z = h.*randn(N,N);
F = exp(-((X.^2+Y.^2)/(clx^2/2)));
f = 2/sqrt(pi)*rL/N/clx*ifft2(fft2(Z).*fft2(F));
surf(X,Y,f,'FaceColor','interp',...
     'EdgeColor','none',...
     'FaceLighting','phong')
axis equal off
mg=[f;Y;X]
es=rot90(mg)
sp=reshape(es,[],3)
csvwrite('E:/rough.xyz',sp)
type E:/rough.xyz

```

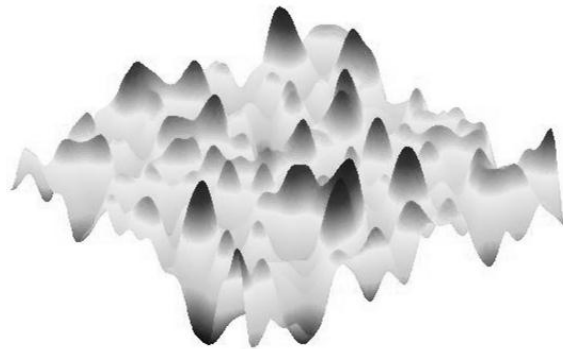


Figure 3.9: Graphic plot of rough surface with $n=500$, $rl=2$, $h=0.5$, and $clx=0.2$.

The surface geometry may be changed by modify the script as shown in Figure 3.10 which the surface is designed as sinusoidal surface. Figure 3.10 is using script below:

```

N=500;
rL=10;
h=0.5;
clx=0.01;
x = linspace(-rL/2,rL/2,N);
y = linspace(-rL/2,rL/2,N);
[X,Y] = meshgrid(x,y);

```



```

% Z = h.*randn(N,N);
% F = exp(-((X.^2+Y.^2)/(clx^2/2)));
% f = 2/sqrt(pi)*rL/N/clx*ifft2(fft2(Z).*fft2(F));
f = h.*cos(5.*X).*cos(5.*Y)
surf(X,Y,f,'FaceColor','interp',...
'EdgeColor','none',...
'FaceLighting','phong')
axis equal off
mg=[f;Y;X]
es=rot90(mg)
sp=reshape(es,[],3)
csvwrite('E:/sinusoidal',sp)
type E:/sinusoidal.xyz

```

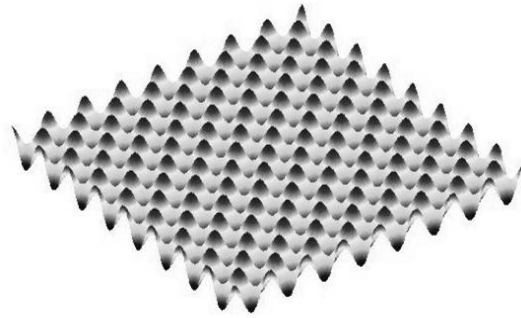


Figure 3.10: Sinusoidal rough surface in “xyz” data format.

The generated surface is converted into coordinate surface data. The raw data is exported from the surface data in a triplet or “xyz” data formats which lists the x, y and z coordinate for each point. The triplet information is reformatted so the surface data file becomes a file which creates and fills a 2D surface array.

3.4.1.2 Surface Modification on SolidWorks

We use SolidWorks to transform coordinate surface into solid surface through connecting each coordinate from surface array. “xyz” data file format is then opened on SolidWorks. It will regard “xyz” file as an array of nodal which shaped like a surface named called point clouds (Fig. 3.11).

At the beginning on SolidWorks, we transformed point cloud into mesh by connecting each nodal to establish rigid geometry of surface. Until this step, the surface was not completely smooth and nearly sharp on each nodal. Therefore, the next step was smoothing the surface with “Scan to 3D” tools in SolidWorks. However, this features only available on “Premium Office” series. This method will generate mesh into solid surface and change its geometry to be perfectly smooth as shown in Figure 3.12.

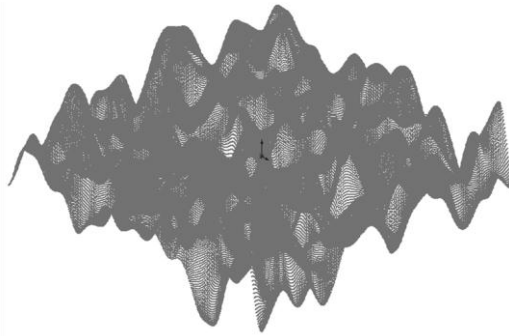


Figure 3.11: Point cloud file in SolidWorks consisted of numerous nodal as a form of rough surface with 0.04 pixel size.

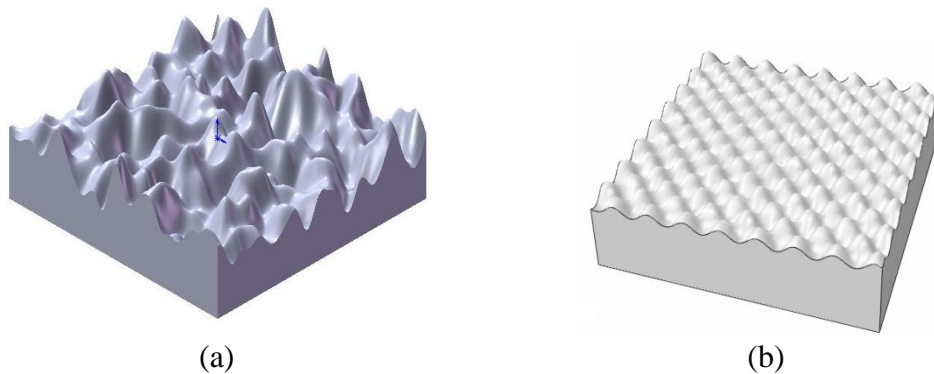


Figure 3.12: Surface which smoothed and given thickness on its surface (a) random rough surface (b) sinusoidal rough surface.

In solid/surface creation during generating process, automatic creation provides better control the shapes of faces and result smoother surface than guided creation. (As long as the number of nodal in “xyz” is high enough). In the end, thickness is given to surface and made it looks likely as rough surface. Finally, the surface is saved in IGES

(initial graphics exchange specification) format data. It is a file format which defines a vendor neutral data format that allows the digital exchange of information among computer-aided design (CAD) systems.

3.4.2 Process pre-processing from ABAQUS

a) Part

Part geometry is used to create model before simulation. Geometry can be directly made in ABAQUS and can also imported from the CAD software. Rough surface geometry here is obtained from the CAD software format with the extension (*.IGES). Geometry making procedures are shown in Figure 3.13.

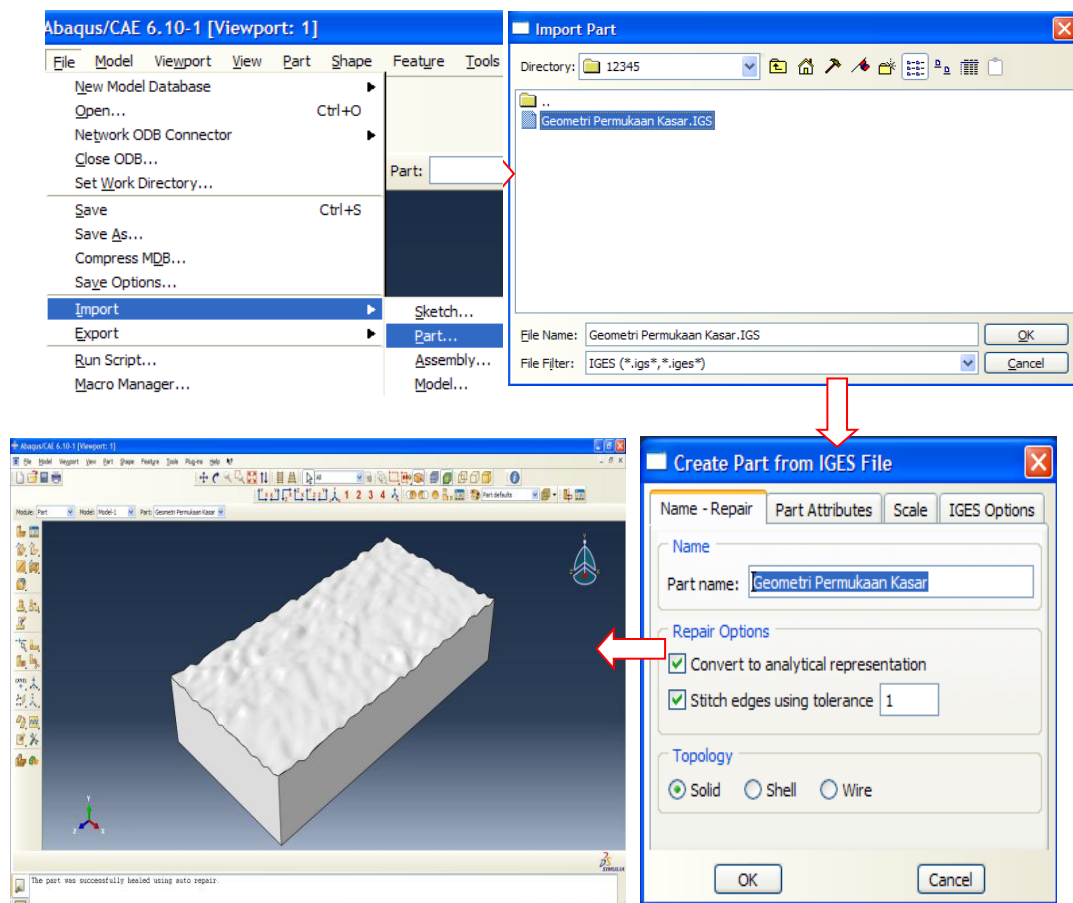


Figure 3.13: Import geometry procedures on ABAQUS.

Rigid spherical geometry is directly created on ABAQUS. The procedures of making rigid spherical geometry are shown in Figure 3.14.

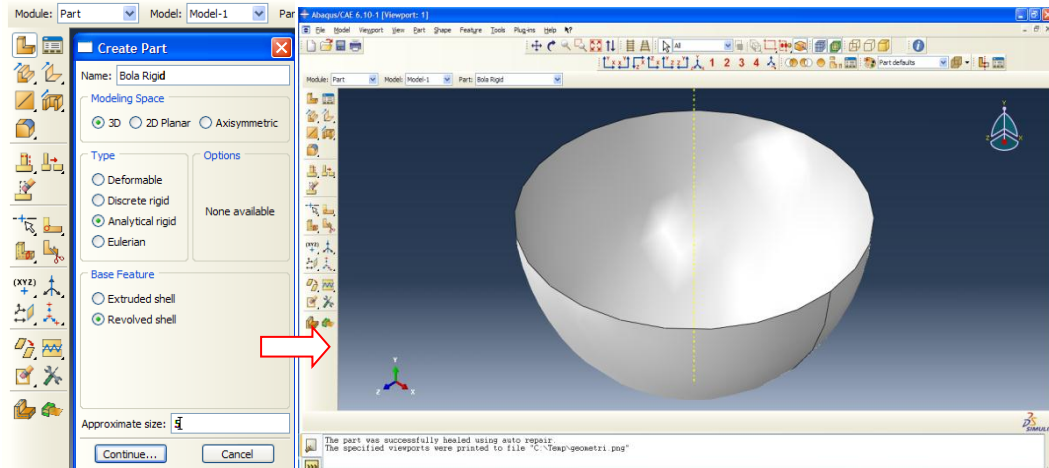


Figure 3.14: Procedures of making rigid ball geometry.

b) Property

Geometry that has been made on the part is then given material properties. The material used here is elastic-perfectly plastic material with Young's modulus of aluminum $E = 75.2$ GPa, Poisson's ratio $\nu = 0.34$ and the yield stress $\sigma_y = 85\,752$ MPa [1]. The material properties when plotted in the form of stress-strain graph shown in Figure 3.15.

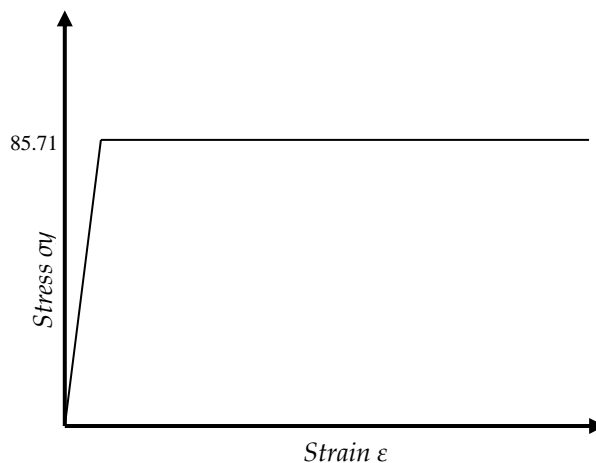


Figure 3.15: Graphic of aluminum alloy material properties [1].

Material properties graphic of aluminum alloy then used in table stress-strain as in Table 3.1.

Table 3.1: Stress-strain of aluminum

Yield Stress	Plastic Strain
85.714	0
85.714	0.005
85.714	0.01
85.714	0.015
85.714	0.02
85.714	0.025
85.714	0.03
85.714	0.035
85.714	0.04
85.714	0.045
85.714	0.05

The values of material properties are then submitted into software ABAQUS as the procedure in Figure 3.16.

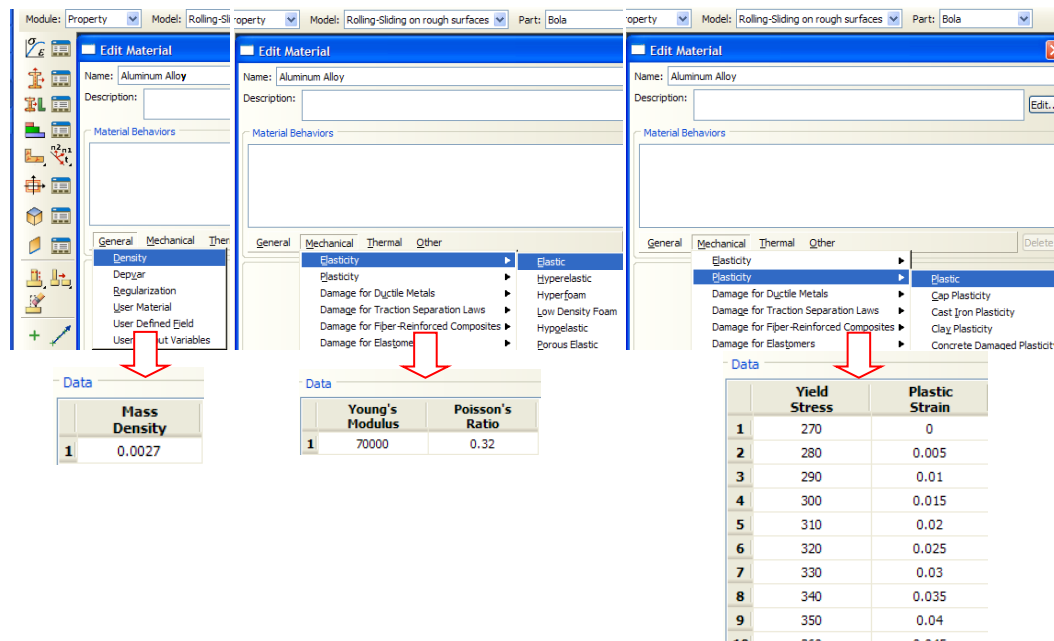


Figure 3.16: Material data input procedures.

Determining material properties of the rough surface geometry is necessary, while for the rigid spheres is other hand because the nature of the analytical rigid does

not require material properties. Procedures of applying material to geometry are shown in Figure 3.17.

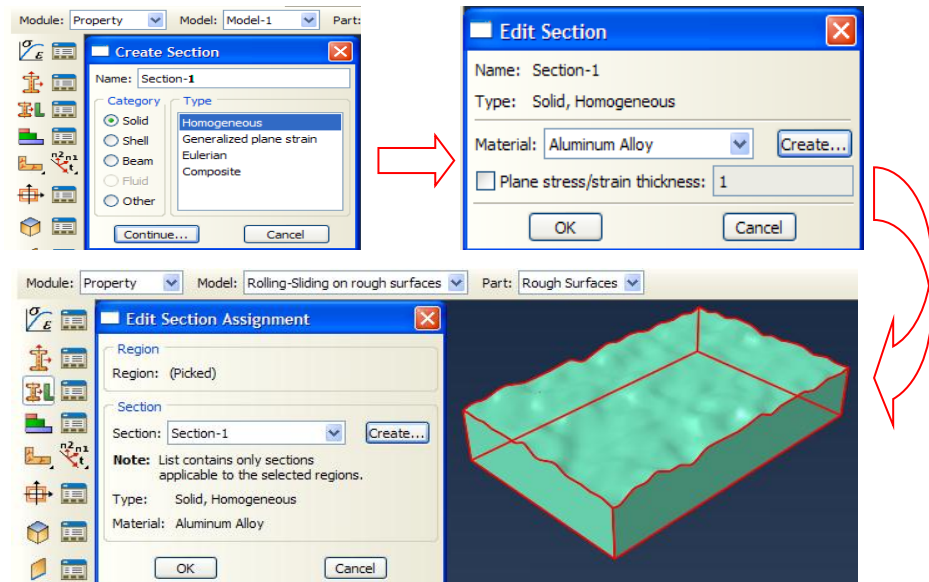


Figure 3.17: Procedures of applying material to geometry rough surface.

c) Assembly

Part geometry at this stage is rough surfaces and spherical geometry rigid. Two geometries are then strung together (assembly) into a screen like the one in the assembly procedure in Figure 3.18.

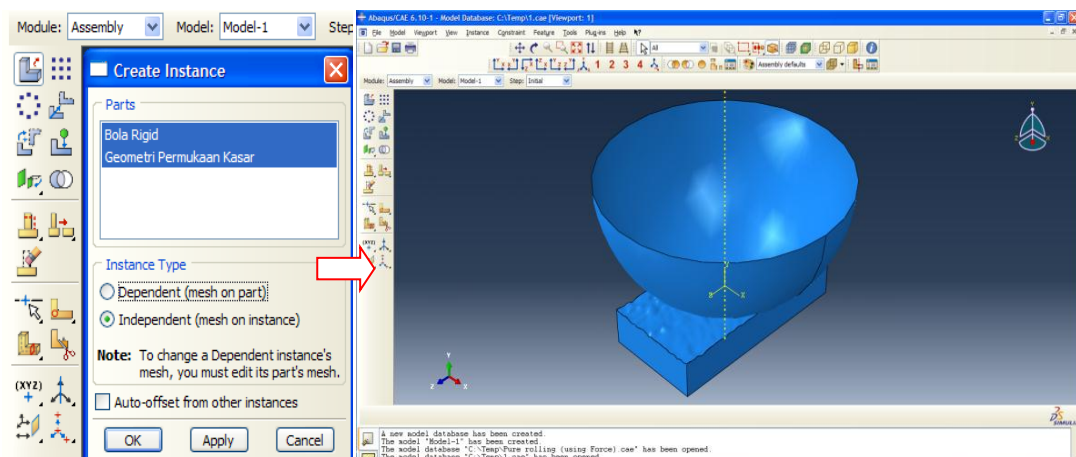


Figure 3.18: Procedure of assembly.

d) Step

Step procedures is to determine the type of analysis carried out in the ABAQUS software. This type of analysis carried out in this study is a static analysis. Step procedure as in Figure 3.19.

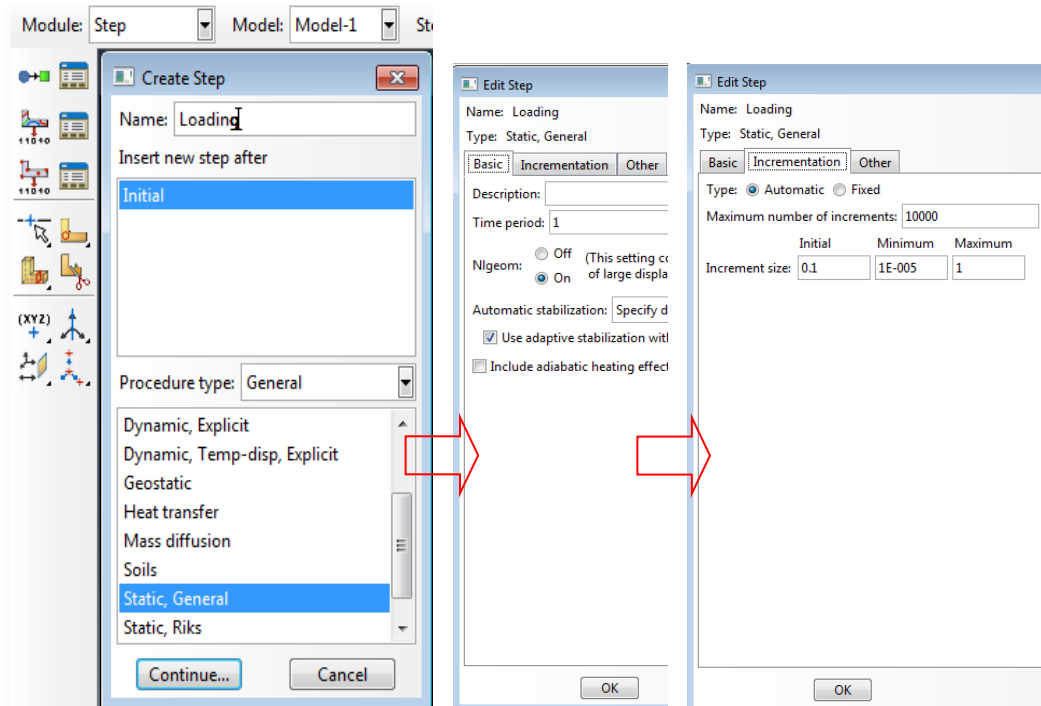


Figure 3.19: Procedures of step.

e) Interaction

Interaction is used to define the interaction between the contact surfaces. Contact interaction in this case is the interaction between rigid spherical surfaces with a rough surface. Rigid spherical indenter surface as it is defined as the master surface, while the rough surface as the slave surface. Contact interaction that occurs has a friction coefficient of 0.3 (Fig 3.20).

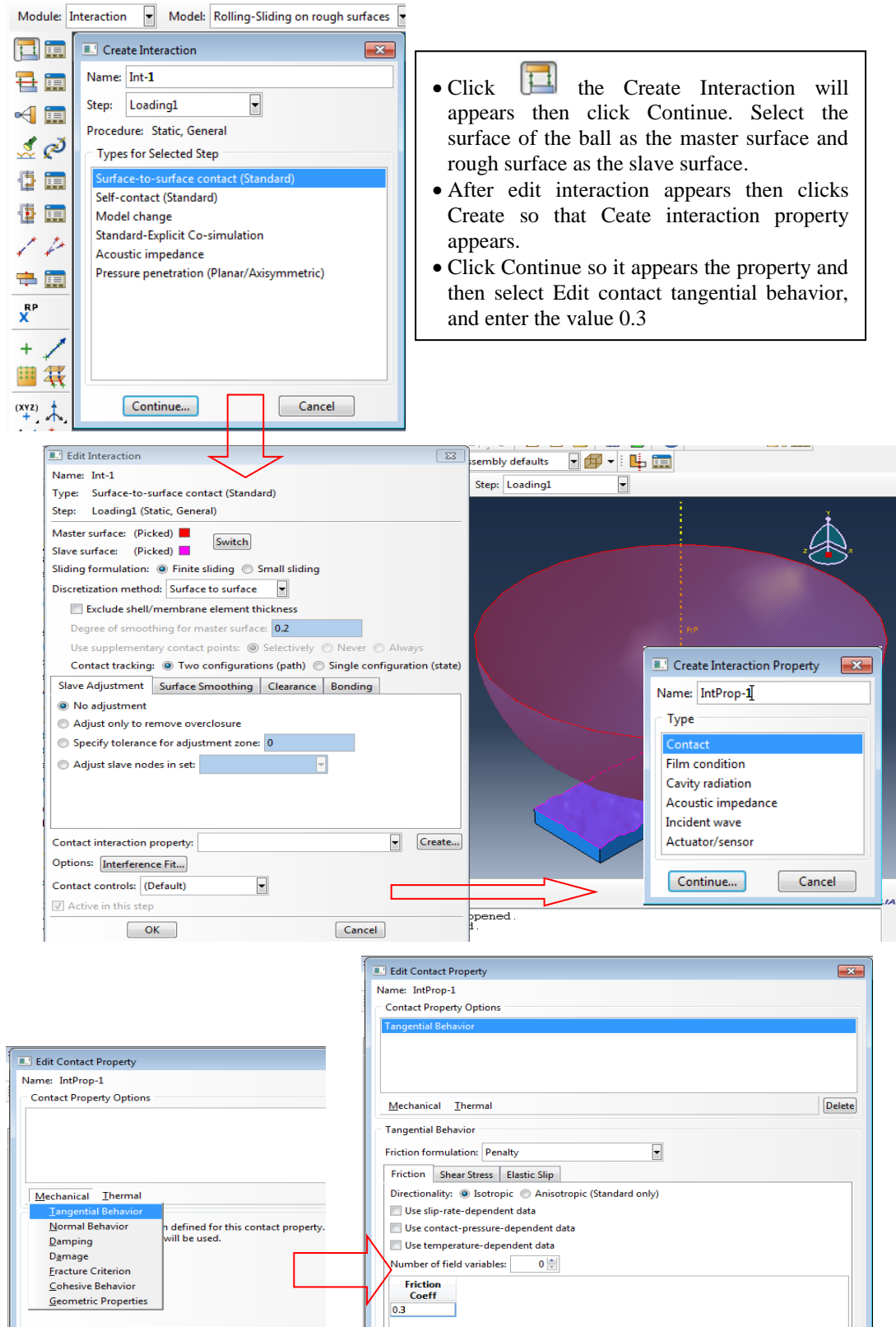
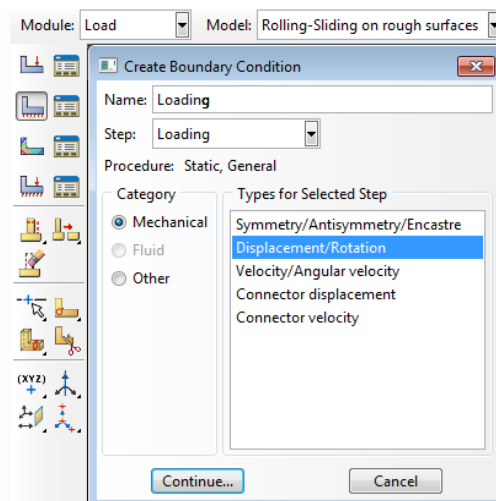




Figure 3.20: Procedures of interaction properties.

f) Load

Load is necessary to determine the loading and boundary conditions (constrain). Load can be a force, pressure, moments and so far depending on the type desired load (Fig 3.21). Boundary conditions can be a displacement, rotation, velocity and so far. As a rigid spherical indenter load is either in the form of interference or contact force at the center of the ball depends desired value. Bottom surface of the rough surface geometry given boundary conditions are not able to move toward any load procedure as in Figure 3.22.



- Click  so that Create Boundary Condition tab appear, then click Continue, chooses center point of the ball, click done.
- Then appear Edit Boundary Condition, select as on picture then click OK.
- Click  so that Create Boundary Condition will appears, then click Continue, select the lower surface of the rough surface geometry, klik Done.
- Select as on picture then click OK. Edit Boundary Condition will appear, select as on the display and then click OK.

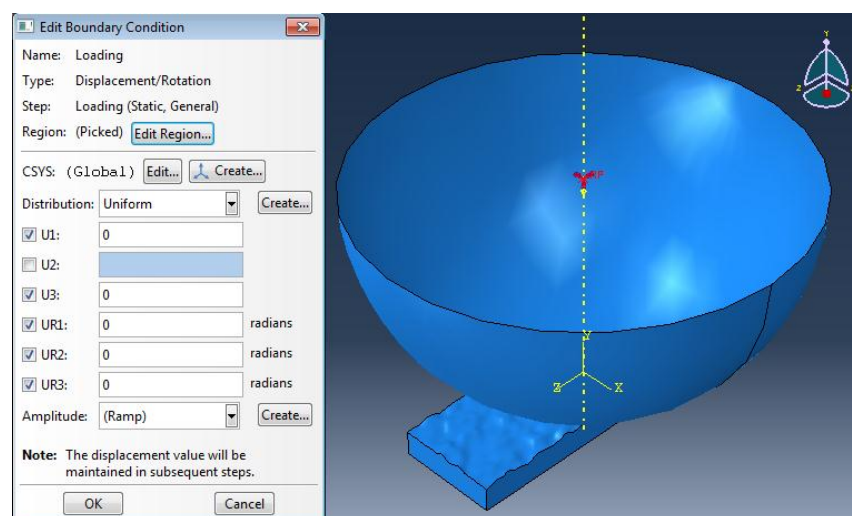


Figure 3.21: Procedures of applying load.

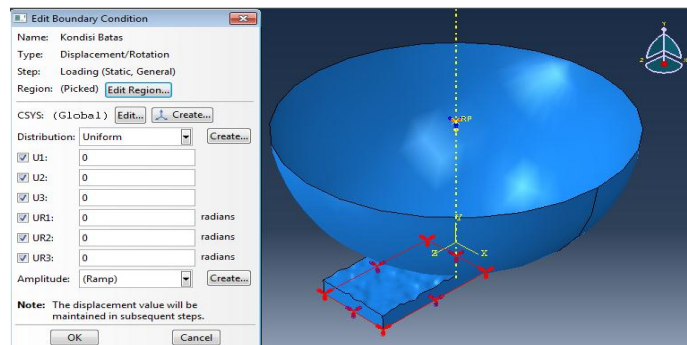


Figure 3.22: Set-up boundary condition.

g) Mesh

Mesh is to discretize geometry to several elements. Types of elements used C3D6 (A 6-node linear triangular prism). Mesh procedures as in Figure 3.23.

I. Global seeds.

Entire surface sized 0.1 length /elemen.

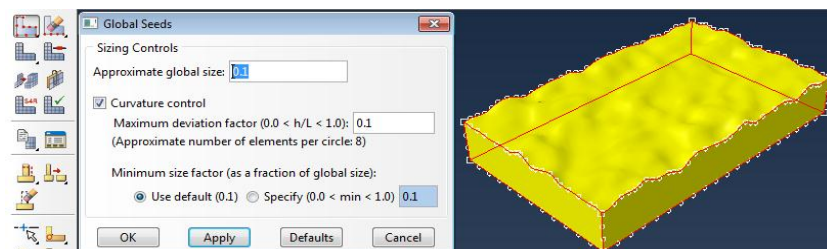


Figure 3.23: Sizing control.

II. Local seeds.

Giving the size on rough surface with the length 0.1/elemen (Fig 3.24).

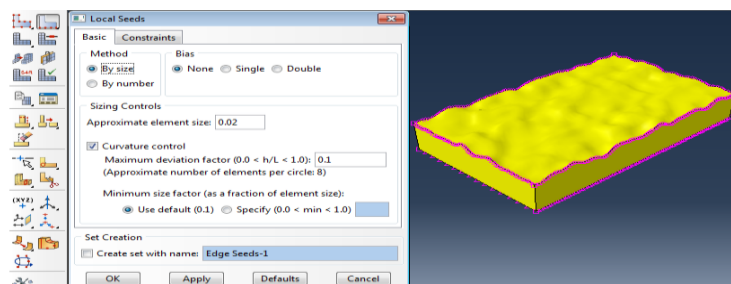


Figure 3.24: Sizing control on local area.

III. Mesh Controls

Select wedge (C3D6) as types of element shape (Fig 3.25).

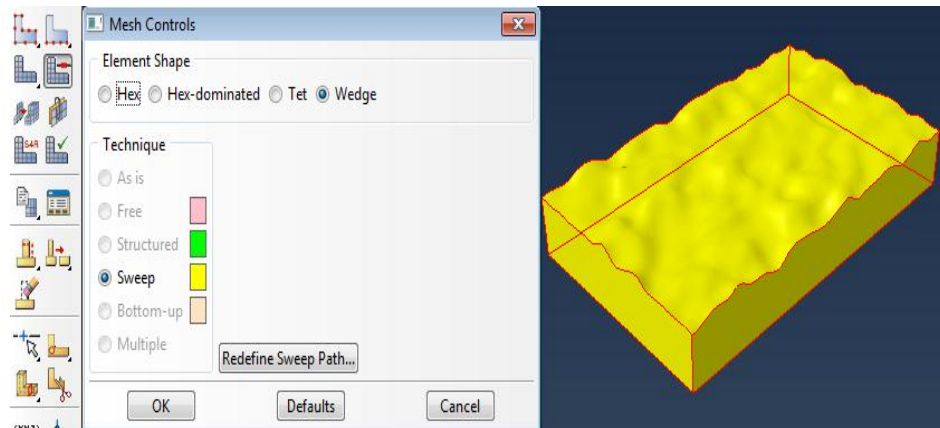



Figure 3.25: Element shape control.

IV. Mesh part instance

Click  to run mesh process then the result as on the picture below (3.26).

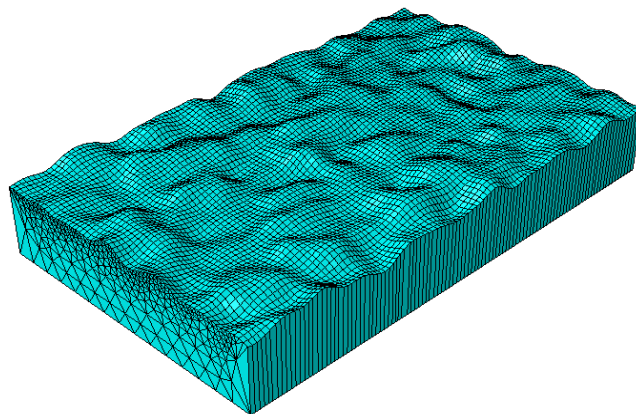


Figure 3.26: Surface after being meshed.

h) Job

Job is used to order ABAQUS iterating or calculating several step that have been set before. Determining memory and number of processor are necessary to optimize time required during iterating (Fig 3.27).

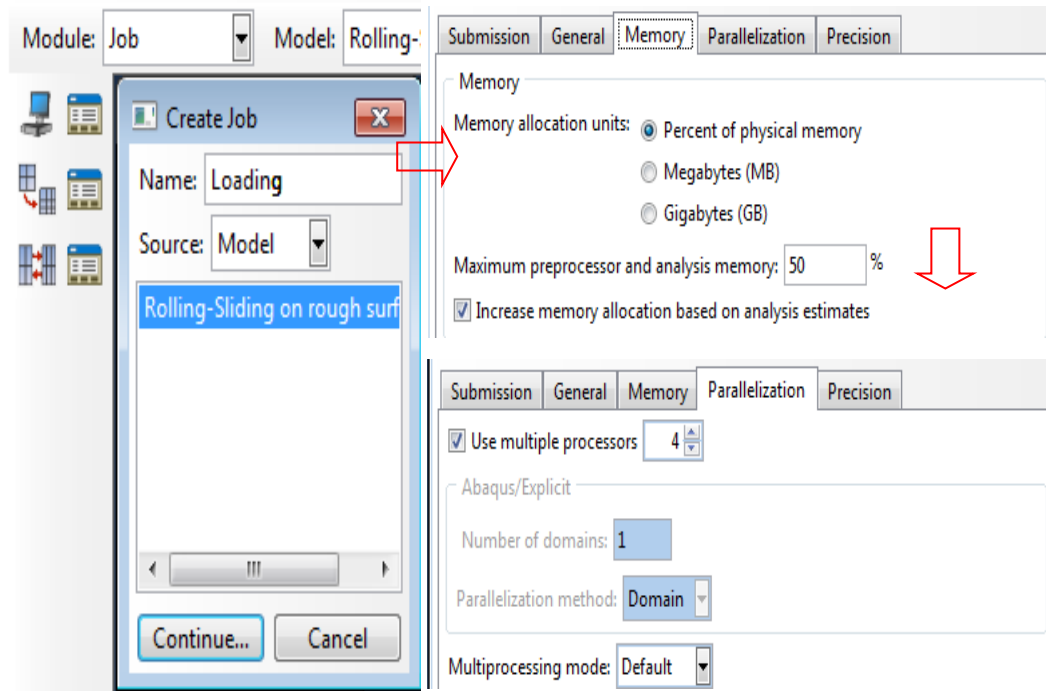


Figure 3.27: Procedure of job.

i) Visualization

Visualization is used to evaluate the results of the simulation. The results of visualization can be a stress contours and deformation. Results visualization will be shown in chapter four in results and discussion.

3.4.3 Verification of Generated Surface

In order to measure whether present method represents condition of real surfaces or not, authors made comparison between two methods of works. The software package ABAQUS was used to analyze the two models created in this study. Quarter sphere elastic contact loaded against rigid plat model was chosen to validate the model. The first quarter sphere model is generated normally in ABAQUS. Meanwhile, the other is generated from present method. Details of the method can be found in Figure 3.28.

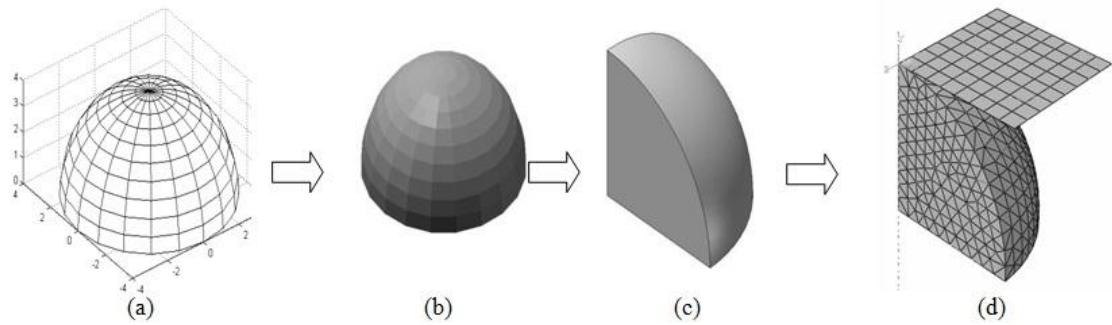


Figure 3.28: (a) Surface generated in form of graphic plot (b) mesh surface after being imported on SolidWorks still has sharp geometry on each nodal (c) half sphere after surface treatment is cut to reduce time computer required (d) surface on ABAQUS.

In the end, the result from those works will be compared to validate present method. In these cases the material properties is considered aluminum ($\nu=0.32$; $E=70$ MPa) with corresponding material model is frictionless. The quarter sphere has 4 mm radius. Contact between the bodies was defined as part of the interaction direction and restrained in the x-direction and the z-direction. The load was defined by applying a constant displacement in the y-direction to the rigid contact plane. This displacement corresponded to the specified interference. For each simulated interference, the model was remeshed to suit the magnitude of deformation expected. These meshed were divided using partitioning tool in the software allowing different meshes of varying element size to be created across the model. Mesh densities were increased around the contact area, and decreased in areas which far from it. An objective in specifying the mesh was to ensure a smooth transition from finely meshed areas to coarser areas thereby avoiding discontinuities in results across partition boundaries within the model.

From the result of simulation, we got contact evolution comparison between contact pressure/S22 with distance from edge of sphere under 35640 element of mesh and 0.1 mm interference (Fig. 3.29). The graphic is coinciding each other indicate that contact pressure on current location of surface is not slightly different. The error from two models is less than 2% with similar contour von Mises stress distribution (Table 3.2 and Fig 3.30). A comparison of results from the elastic simulation during validation showed good agreement. It means that model surface generated from SolidWorks into

ABAQUS and model normally created in ABAQUS are alike. It means further analyzes using present method is justifiable.

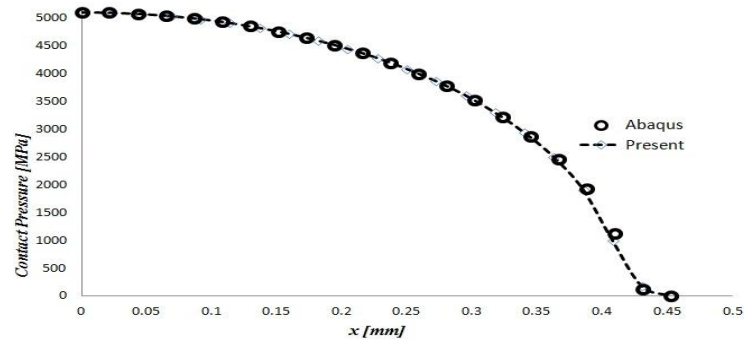
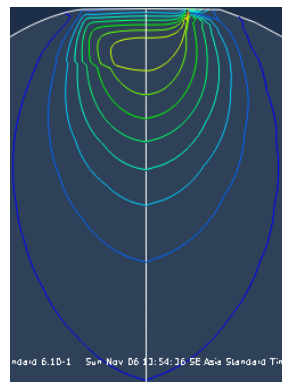


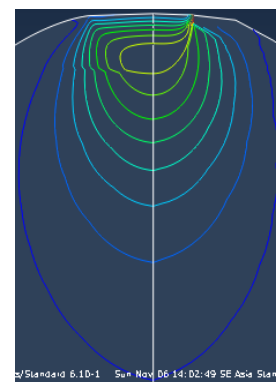
Figure 3.29: Comparison between ABAQUS and present model shows good agreement.

Table 3.2: Comparison of von Misses stress on two models with varying interference under 433 elements

w	S (von Misses)		Error %
	ABAQUS	MSA	
0.05	1835	1832	0.163488
0.1	4628	4620	0.172861
0.15	5143	5145	0.03889
0.2	6738	6736	0.029682
0.25	8136	8139	0.03687
0.3	9201	9212	0.11955



(a)



(b)

Figure 3.30: Contour of von Misses stress (a) present model (b) ABAQUS model.

Reference:

1. Jamari, J., 2006, Running-in of Rolling Contacts, Ph.D. Thesis, University of Twente, Faculty of Engineering Technology.