Sertifikat Penghargaan

Diberikan kepada:

Andy Prabowo, Ph.D.

Atas kontribusinya dalam menyusun dokumen

Tata cara membuat model balok baja canai dingin berlubang pada program Abaqus

Dr. Ir. Usman Wijaya, S.T., M.T.
Direktur PT. DeltaKoni

1 Februari 2024

TATA CARA MEMBUAT MODEL BALOK BAJA CANAI DINGIN BERLUBANG PADA PROGRAM ABAQUS

DISUSUN OLEH:

ANDY PRABOWO

DAN

KELSEN ANDRIAN PRIESTLEY

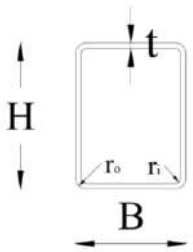
PROGRAM STUDI SARJANA TEKNIK SIPIL

FAKULTAS TEKNIK

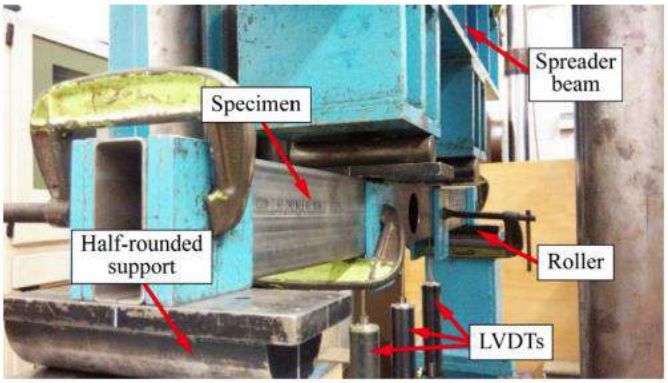
UNIVERSITAS TARUMANAGARA

Februari 2024

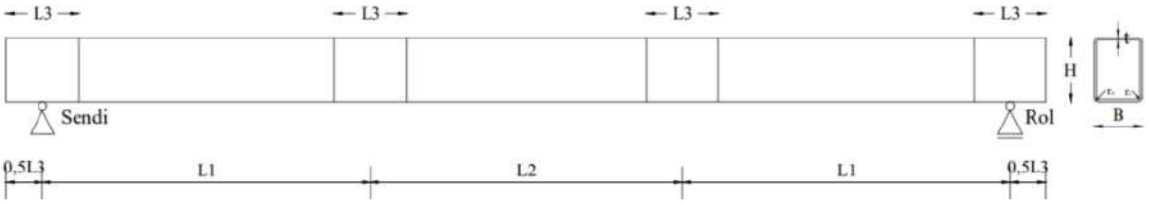
STUDI KASUS



Bentuk Profil *Rectangular Hollow Section* (RHS)



Pengujian *Four-Point Bending* (Chen et al., 2022)



Gambar Tampak Memanjang Dan Melintang Balok RHS

Model yang digunakan adalah RHS *Lean Duplex Stainless Steel* dengan ukuran sebagai berikut:

H (mm)	B (mm)	t (mm)	L1 (mm)	L2 (mm)	L3 (mm)	r_o (mm)	r_i (mm)
120	80	3	410	390	90	6	6

Properti material yang digunakan adalah pada temperatur 900ºC yang didapatkan dari rumus empiris Huang & Young (2014) yaitu sebagai berikut:

Elastic:

E_T = 40259.2308 MPa

$f_{y,T}$ = 54.2724 MPa

$f_{u,T}$ = 66.8872 MPa

$\varepsilon_{u,T}$ = 2.19%

Poisson’s Ratio = 0.3

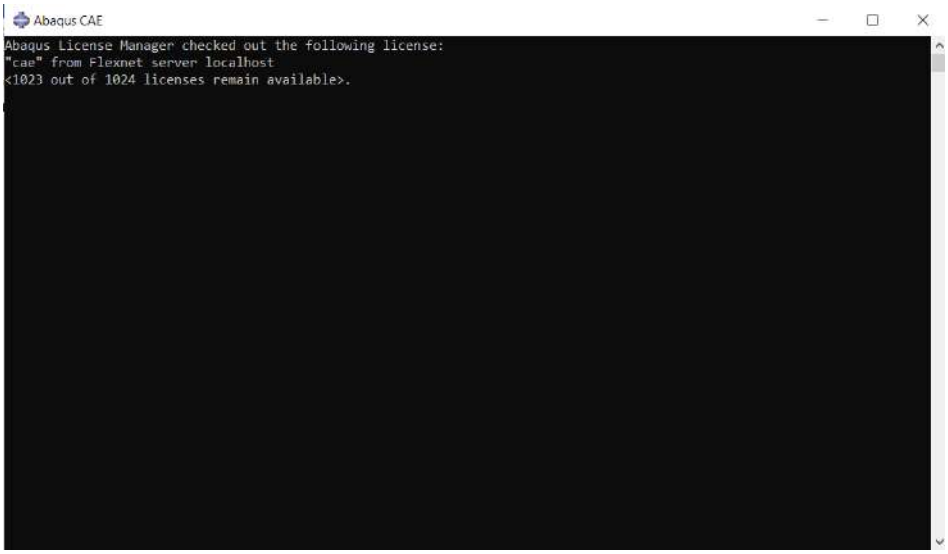
Plastic:

σ_{true} (MPa)	ε_{true}
35.0308	0
45.0598	2.0880E-04
50.0995	7.4324E-04
54.4541	1.9899E-03
56.3705	5.1948E-03
58.6279	9.3111E-03
60.9135	1.3597E-02
63.2244	1.7986E-02
64.3890	2.0209E-02
65.5596	2.2448E-02
66.7360	2.4700E-02
67.9183	2.6964E-02
68.9720	2.8981E-02

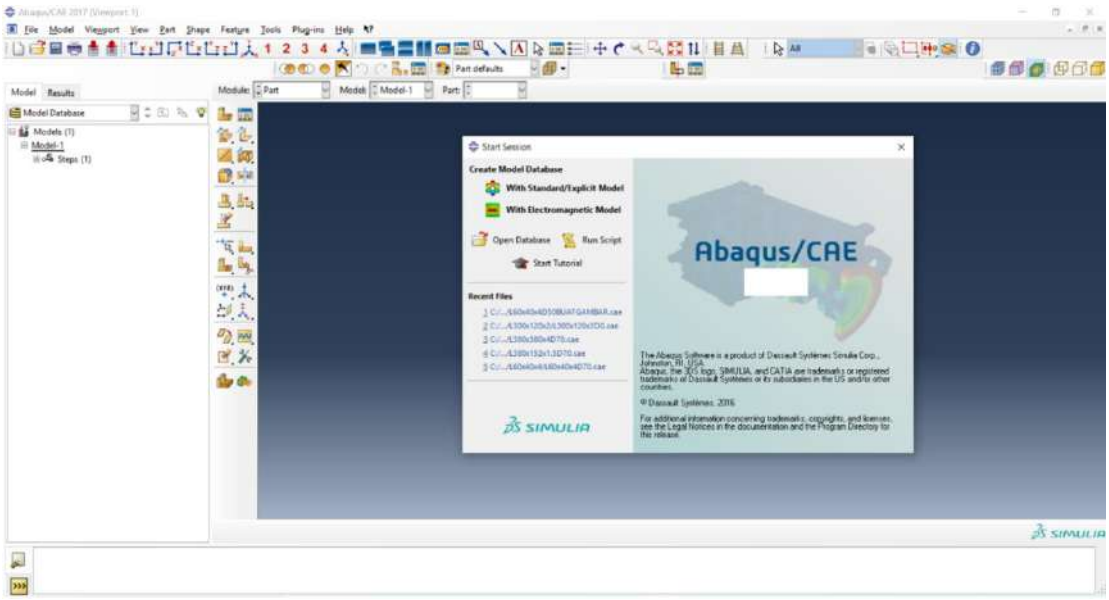
1. Membuka Abaqus.

   Klik *Windows* – Ketik *Abaqus CAE* – Klik *Abaqus CAE*.

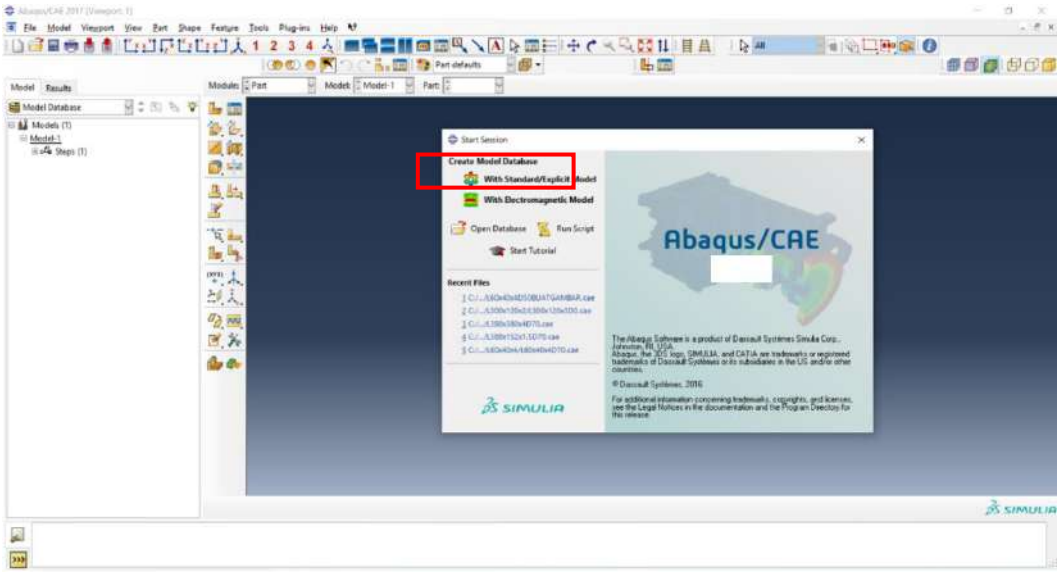
   Pada saat Klik *Abaqus CAE* akan muncul *window Abaqus CAE* seperti dibawah ini.



   Diamkan beberapa saat, lalu akan muncul *window Abaqus/CAE [Viewport 1]*.

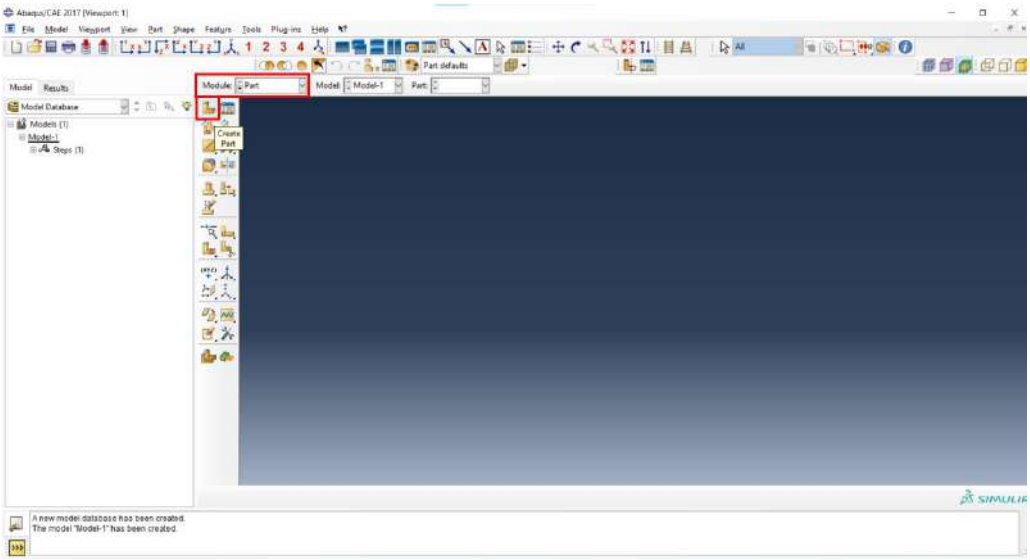


   Pilih *Create Model Database – With Standard/Explicit Model.*

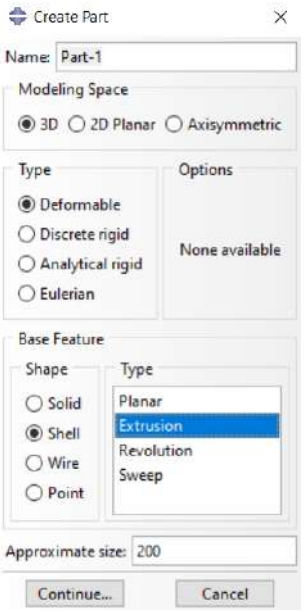


Setelah itu, *window* akan seperti gambar dibawah ini.

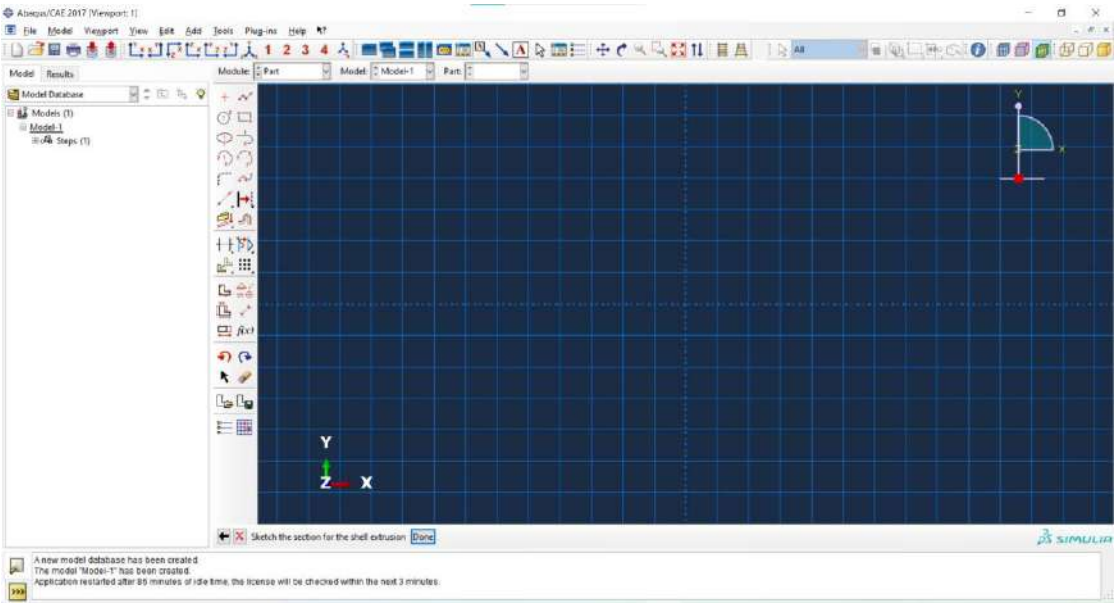
2. Membuat Model, pada *Module: Part* – Klik *Create Part.*



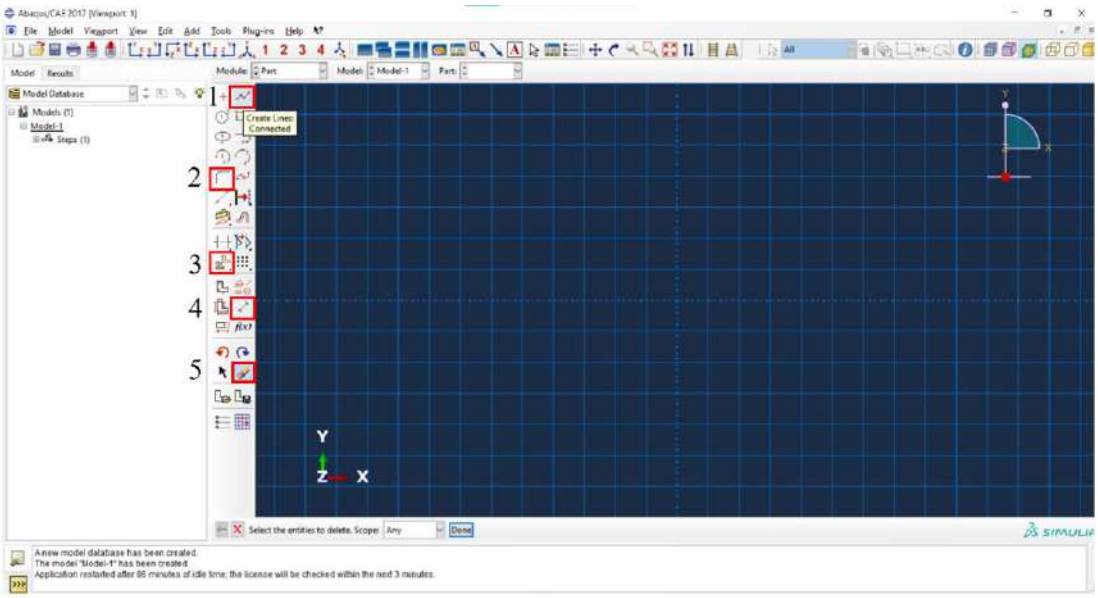
Setelah itu, akan muncul *window Create Part*. Untuk studi kasus ini, digunakan *Modeling Space – 3D, Type – Deformable, Base Feature – Shape – Shell, Type Extrusion* dan *Approximate Size 200.* Klik *Continue…*

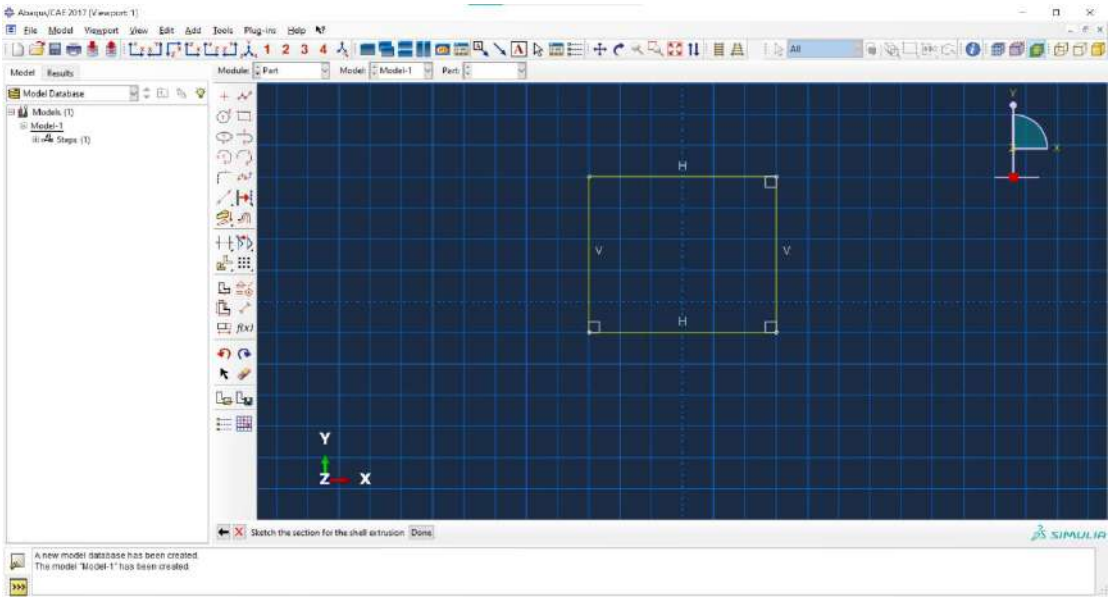


Setelah itu, akan muncul *window* seperti ini.

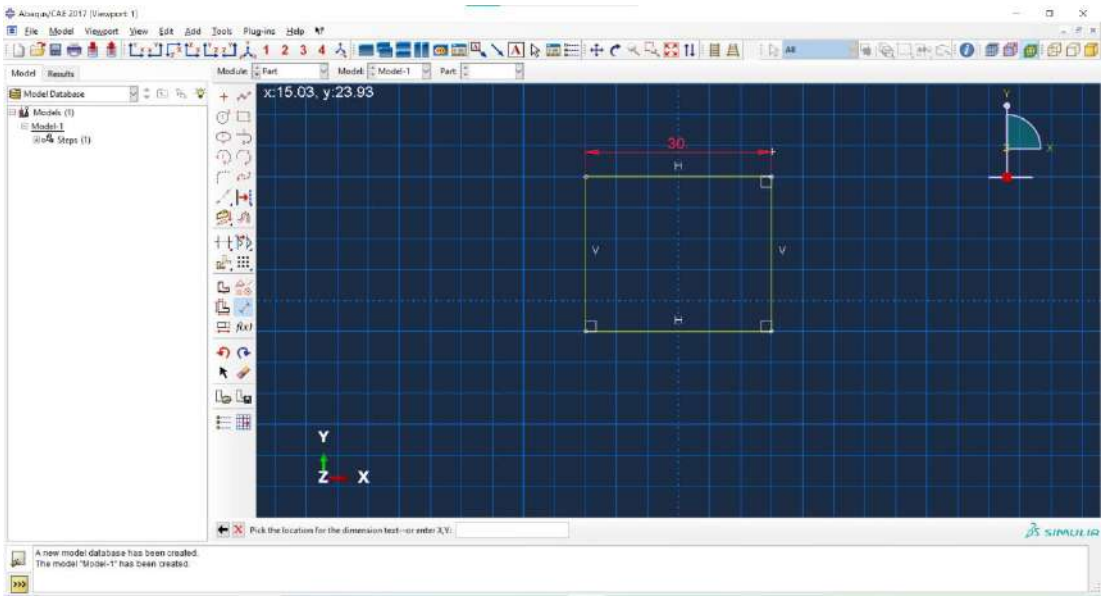


Gunakan *Create Lines: Connected* (1) untuk membuat garis. Gunakanlah tinggi dan lebar bersih serta radius yang diperoleh dari $\frac{(r_o + r_i)}{2}$.

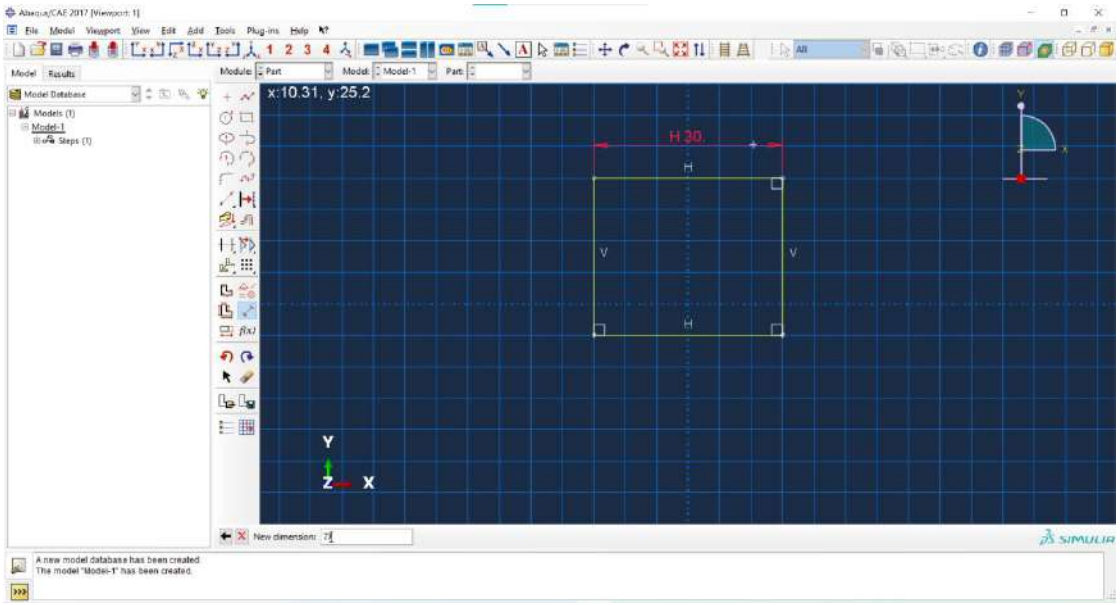


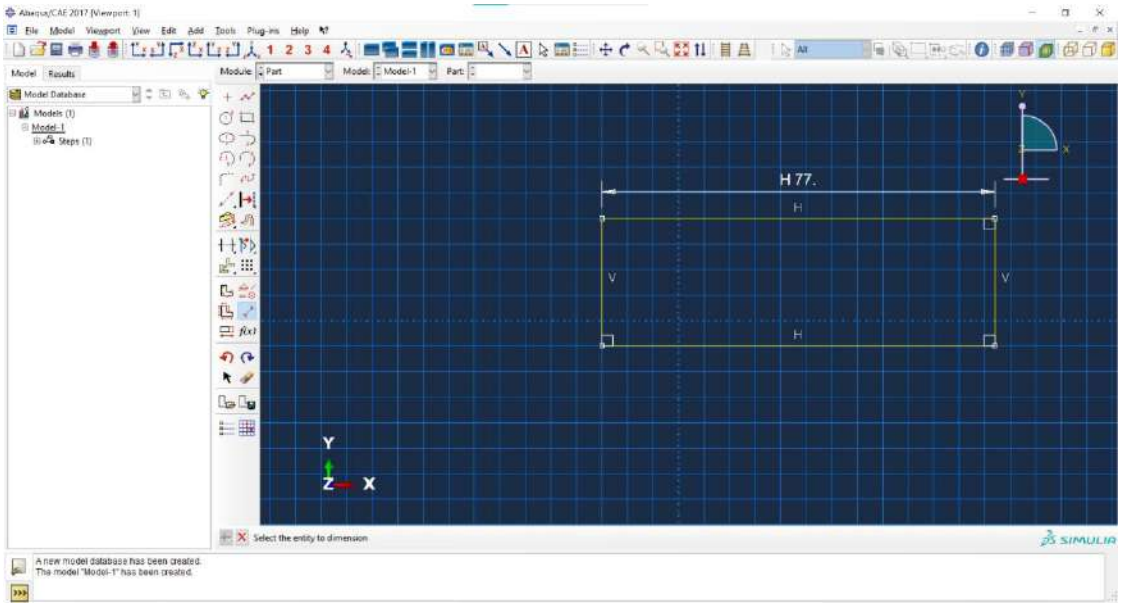


Setelah bentuk kotak sudah seperti gambar, gunakan *Add Dimension* (4) untuk merubah ukuran kotak sesuai tinggi dan lebar bersih. Klik *Add Dimension* (4) – Klik ujung kiri dan ujung kanan salah satu garis lebar yang telah dibuat.

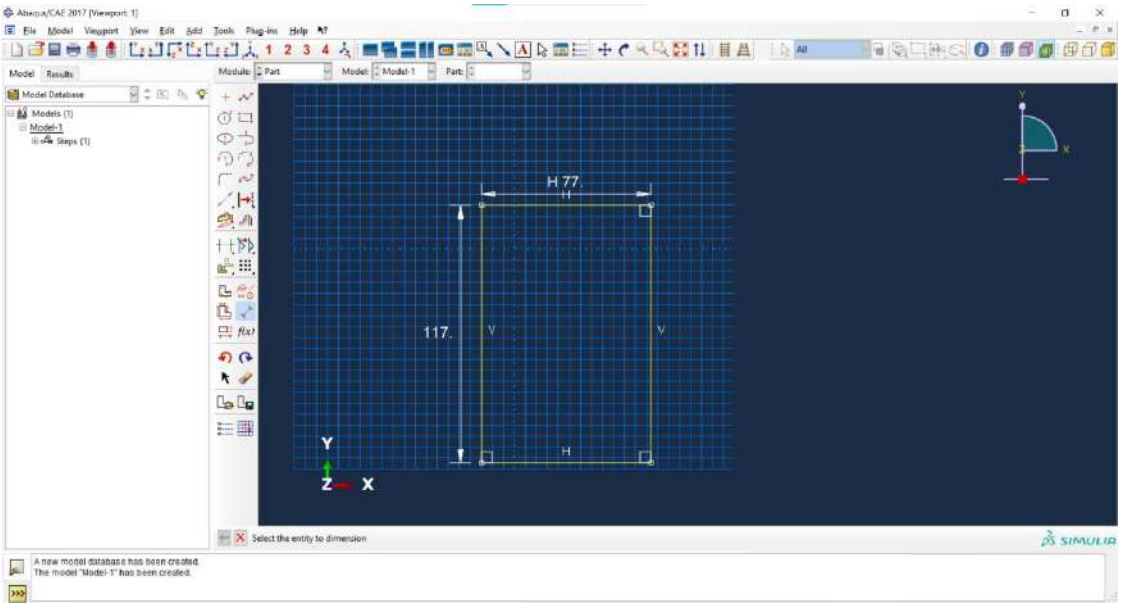


Pada kotak *New Dimension*, ketik lebar bersih profil, yaitu B – t (80 – 3) lalu tekan *Enter*.

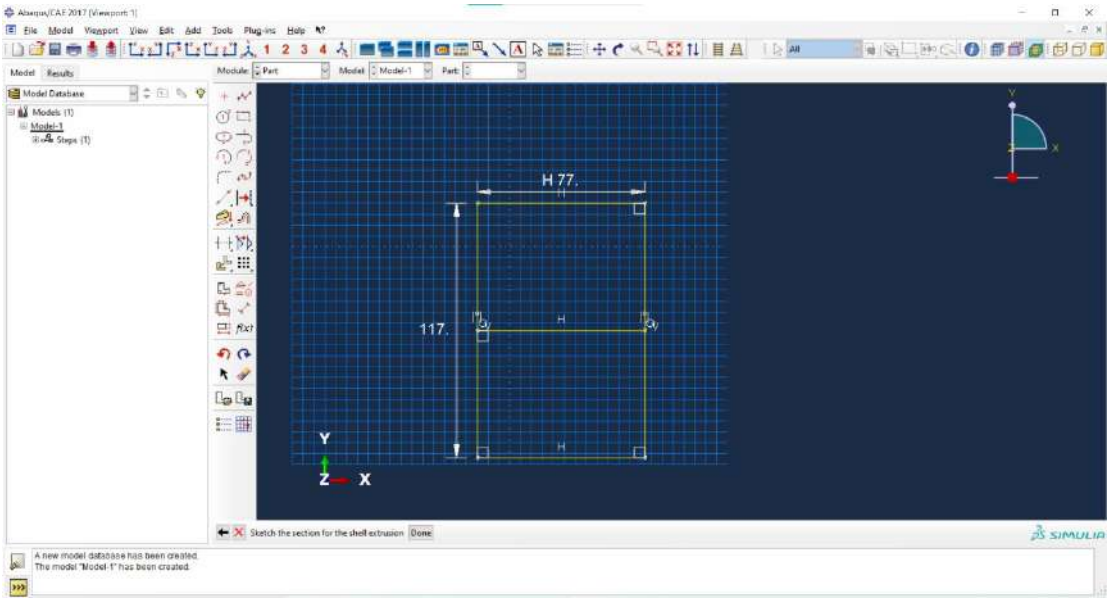


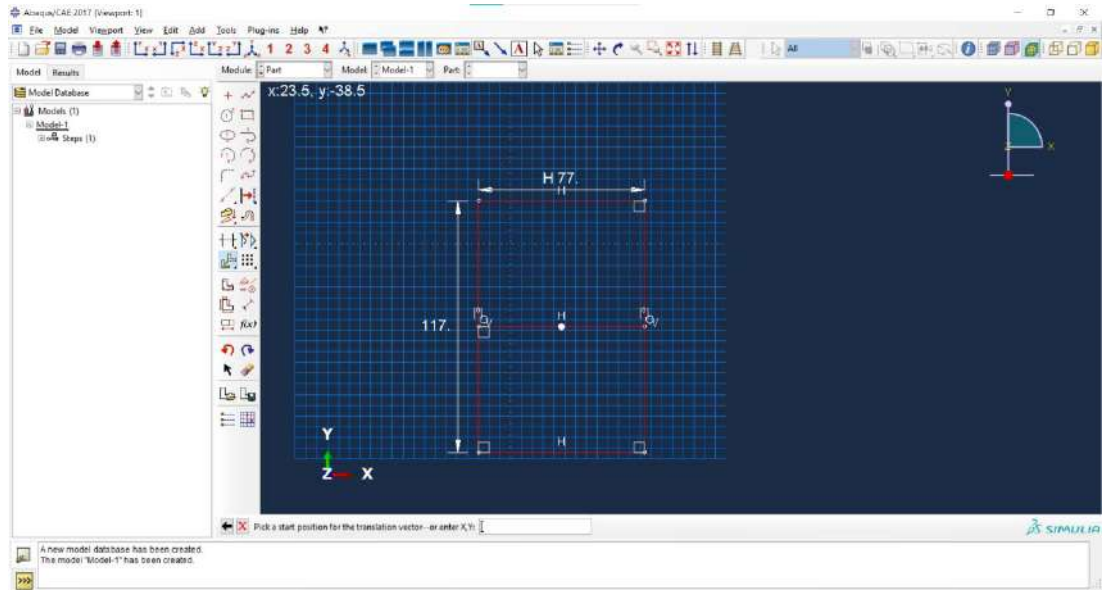


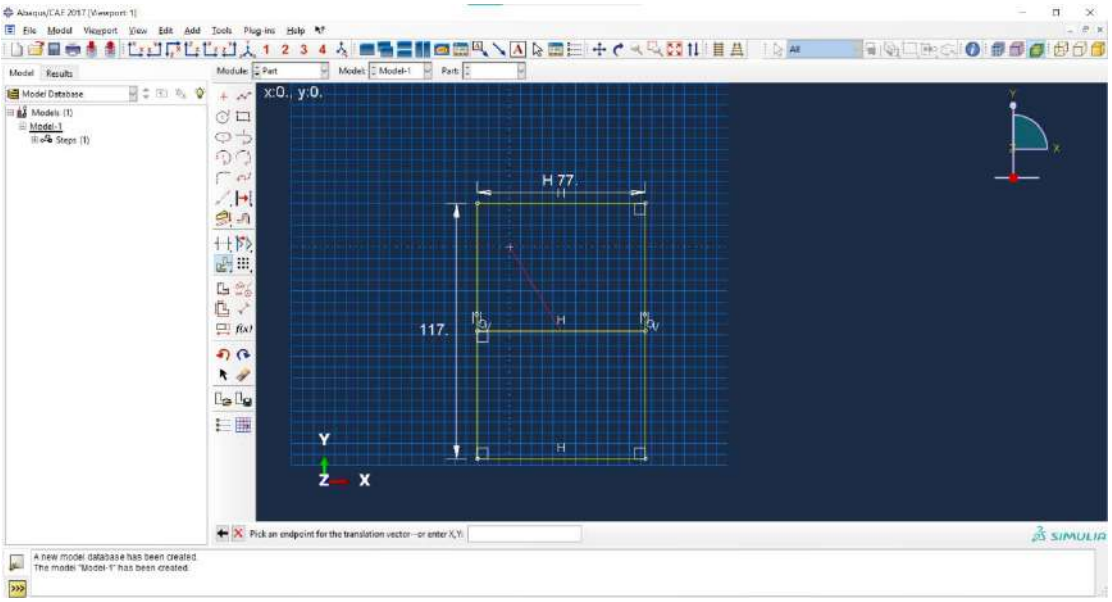
Lakukan hal yang sama dengan garis tinggi sehingga diperoleh bentuk kotak sebagai berikut.

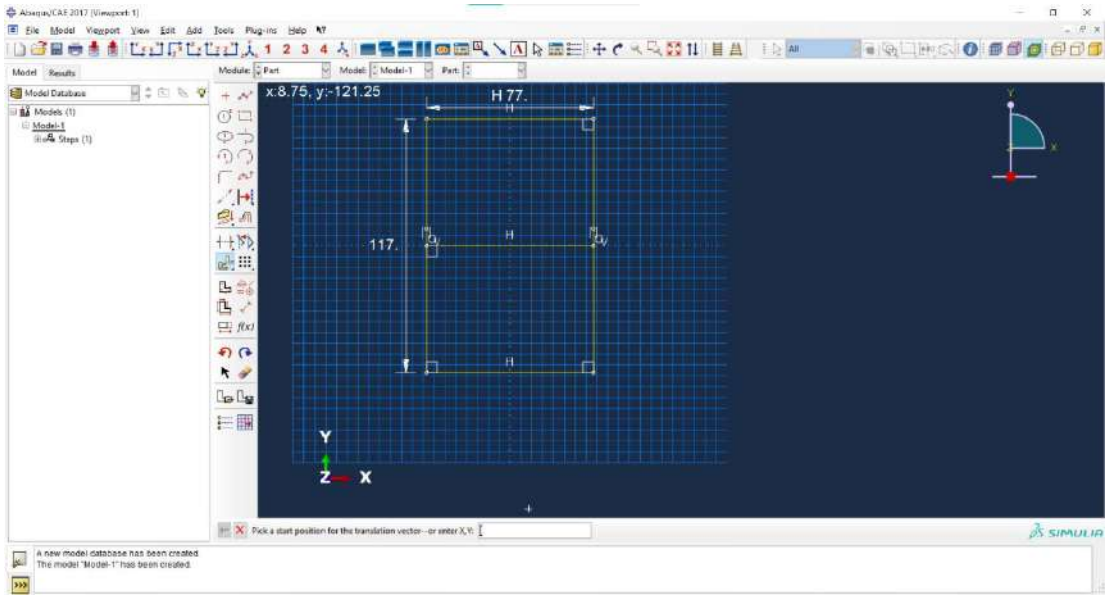


Setelah itu pindahkan bentuk kotak ke titik tengah *grid* dengan menambahkan garis bantu terlebih dahulu. Lalu pindahkan bentuk kotak dengan menggunakan *Translate* (3). Klik *Translate* – *Move* – *Select* bentuk kotak – *Done* – Klik titik tengah garis bantu – Klik titik tengah *grid*. Bentuk kotak akan berada ditengah-tengah *grid.*

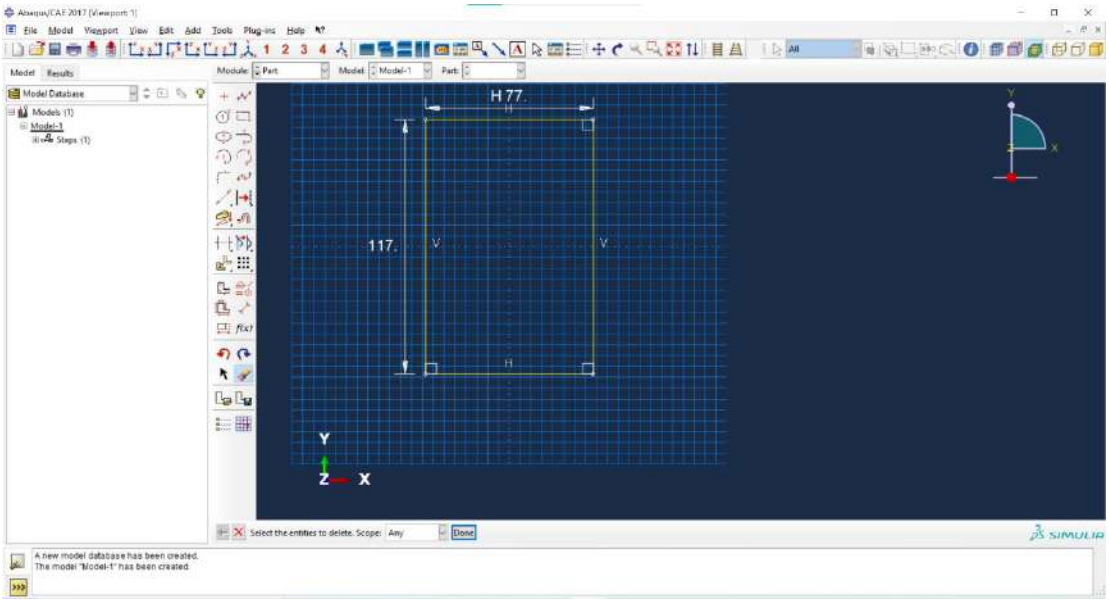



x:23.5, y:-38.5
H 77.
117.
Y
Z
X
Pick a start position for the translation vector--or enter X,Y:
SIMULIA
A new model database has been created.
The model "Model-1" has been created.

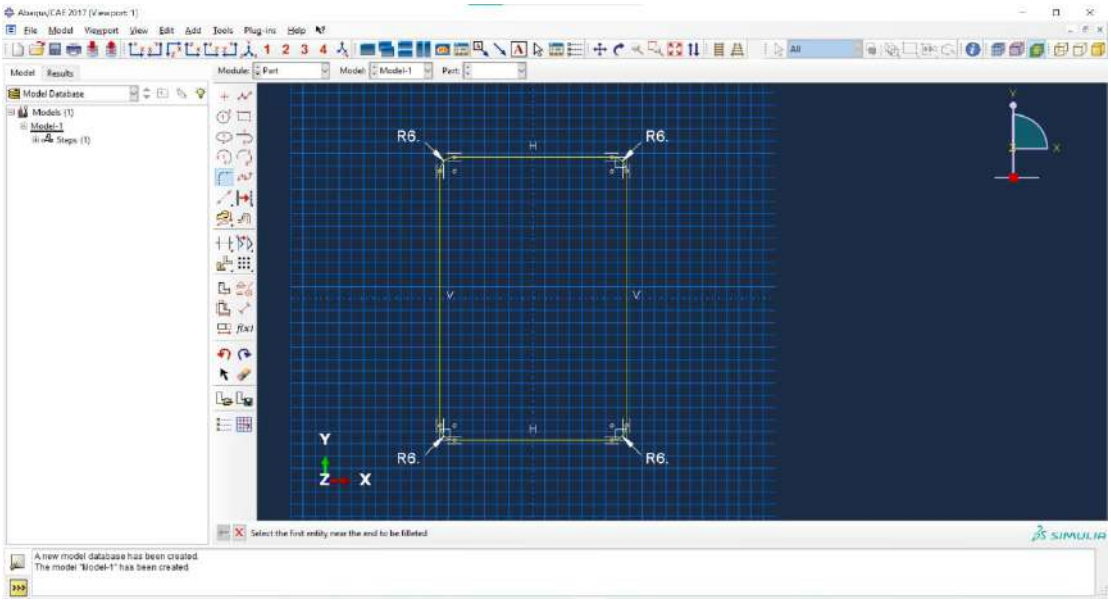

x:0., y:0.
H 77.
117.
Y
Z
X
Pick an endpoint for the translation vector--or enter X,Y:
SIMULIA
A new model database has been created.
The model "Model-1" has been created.


x:8.75, y:-121.25
H 77.
117.
Y
Z
X
Pick a start position for the translation vector--or enter X,Y:
SIMULIA
A new model database has been created.
The model "Model-1" has been created.

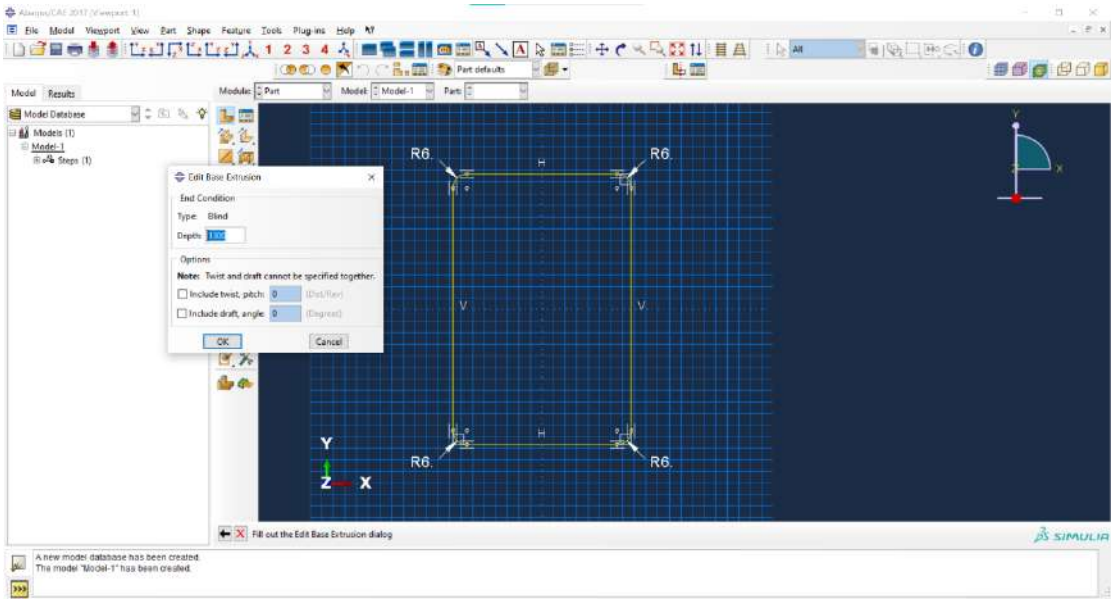
Setelah itu, hapus garis bantu dengan menggunakan *Delete* (5) – *Select* garis bantu – *Done.*



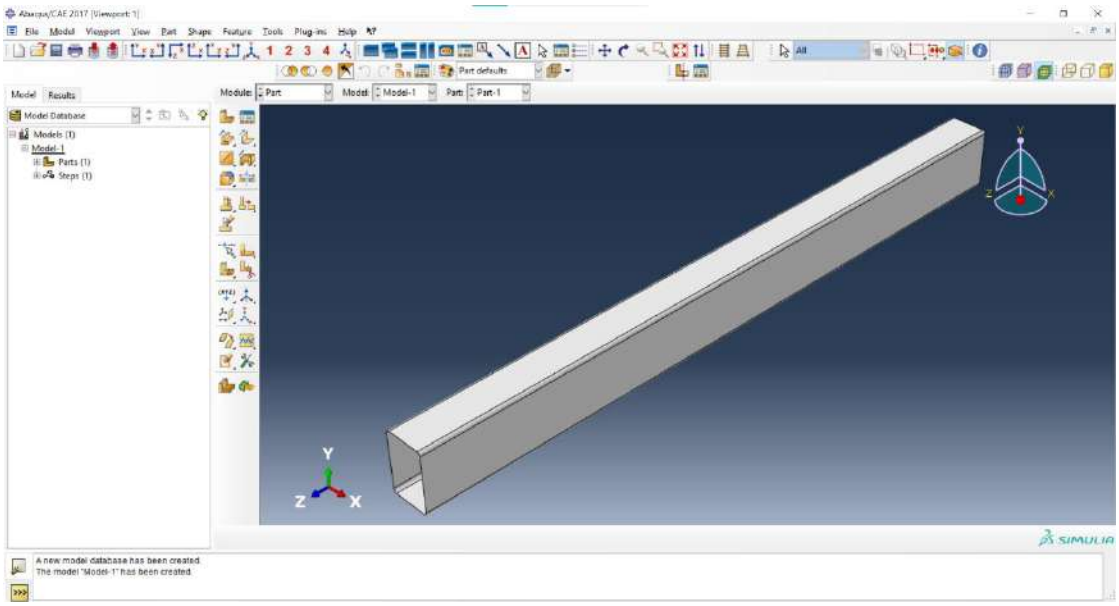
Klik *Create Fillet: Between 2 Curves* (2) – Masukan *Fillet Radius* 6 dari perhitungan $\frac{(r_o + r_i)}{2}$ – *Enter* – Klik garis tinggi dan lebar sehingga terbentuk *fillet.*



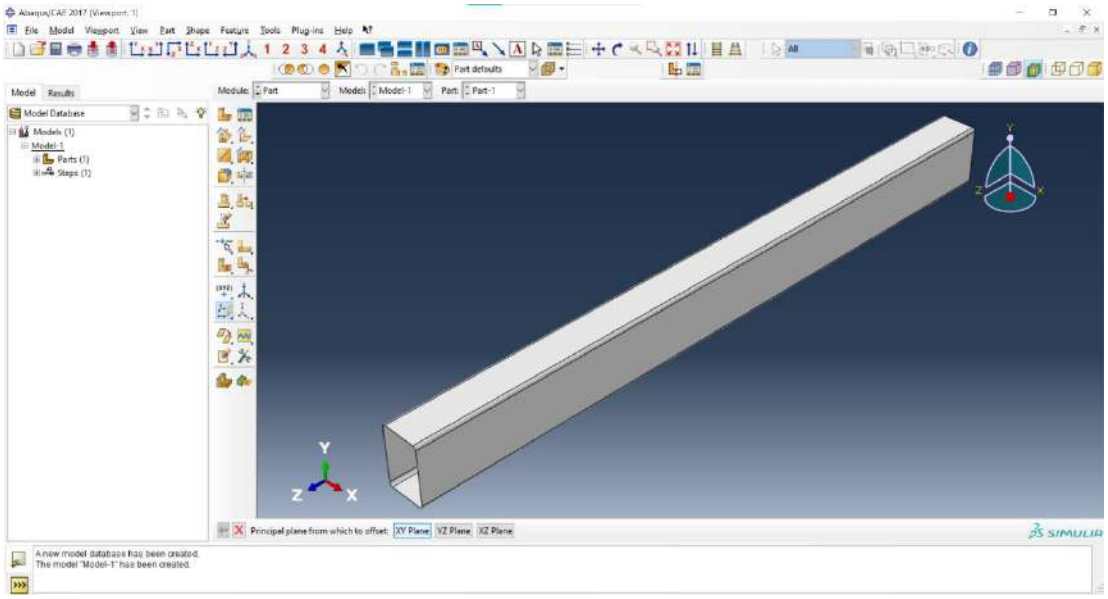
Klik X pada tulisan "*Select the first entity near the end to be filleted*" – Klik *Done* pada tulisan "*Sketch the section for the shell extrusion*".

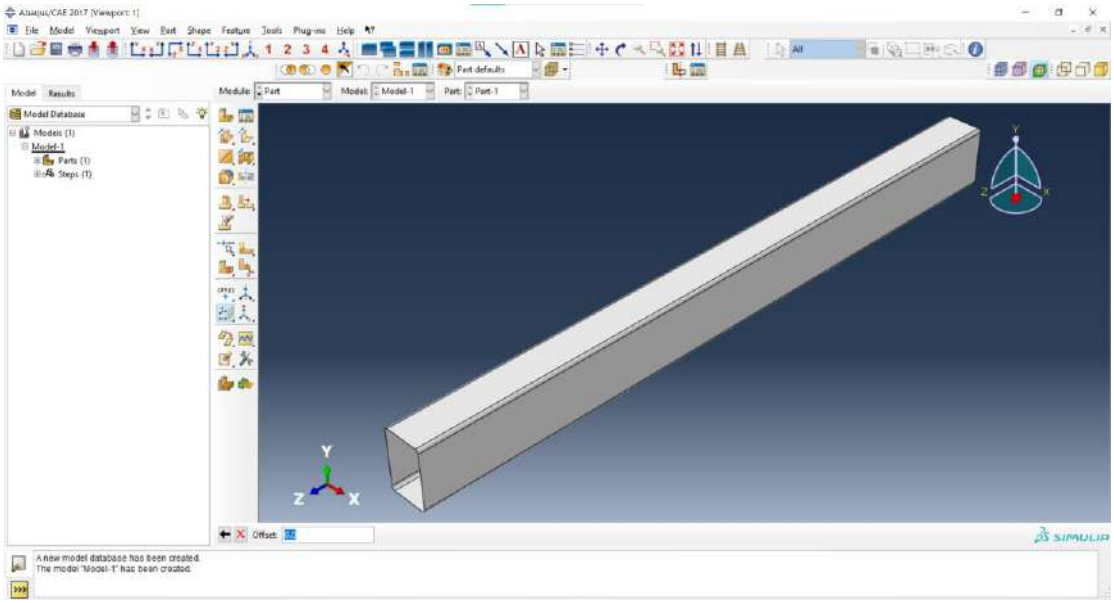


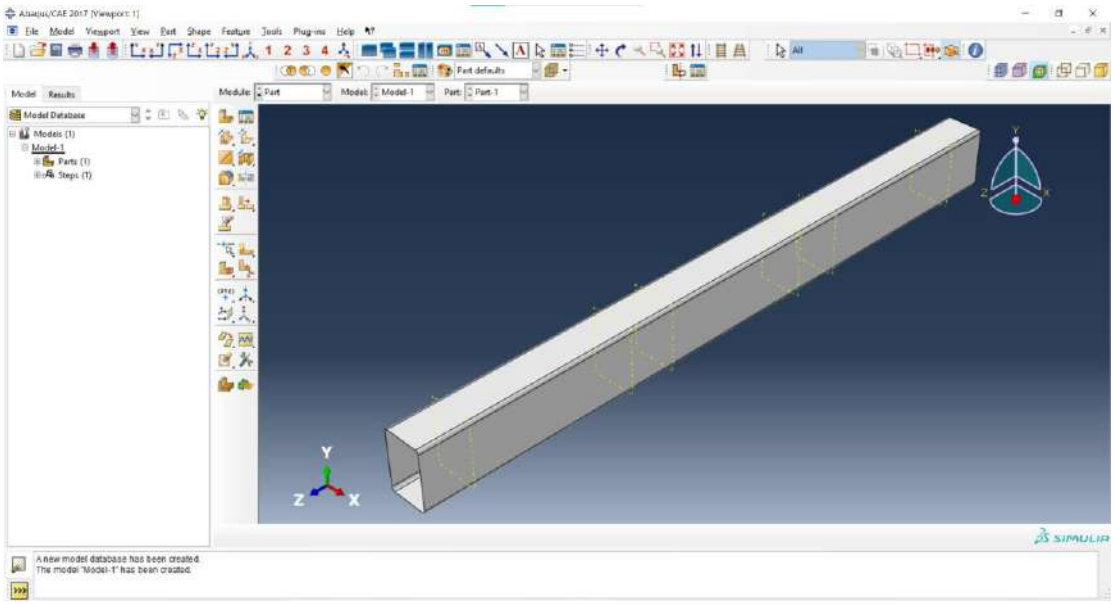
Pada *Edit Base Extrusion – Depth* di isi dengan total panjang bentang model tersebut dimana pada studi kasus ini adalah 1300 – Klik Ok. Maka akan terbentuk model penampang profil RHS dengan panjang total 1300 mm.



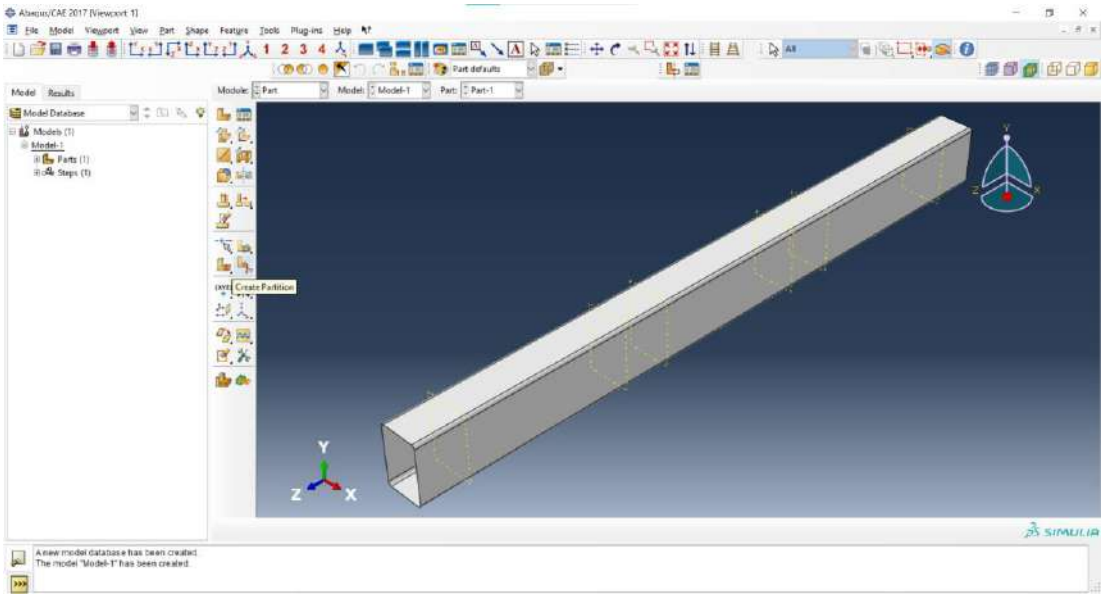
Membuat Partisi, Klik *Create Datum Plane: Offset From Principal Plane* – Klik *XY Plane* – Masukan *Offset* sesuai dengan partisi yang diinginkan – Enter. Pada studi kasus ini, jarak partisi yang diinginkan adalah sebagai berikut: 90-410-500-800-890-1210.

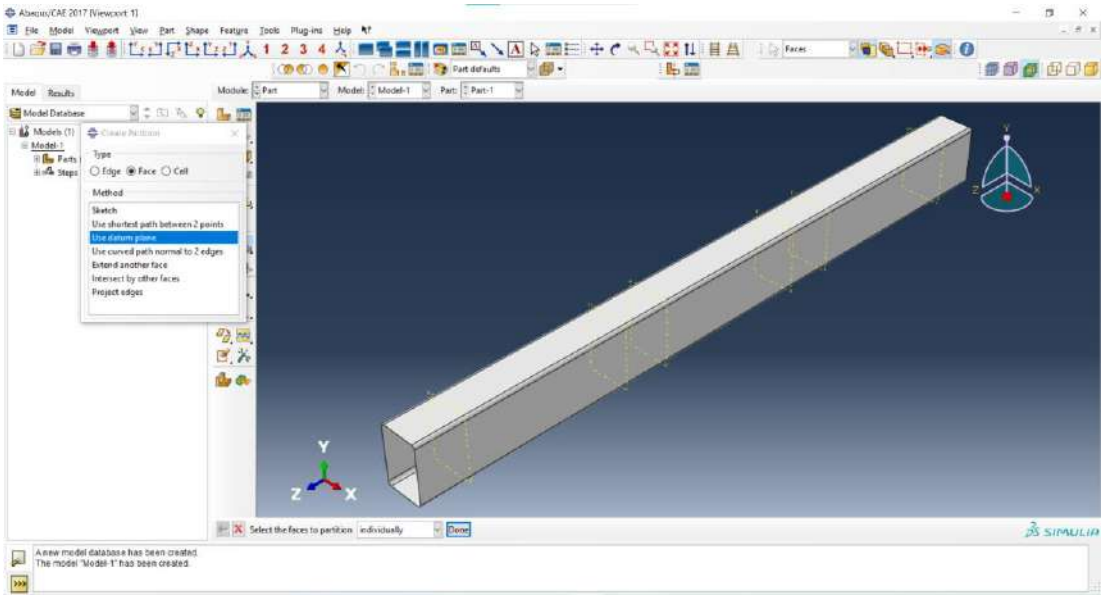


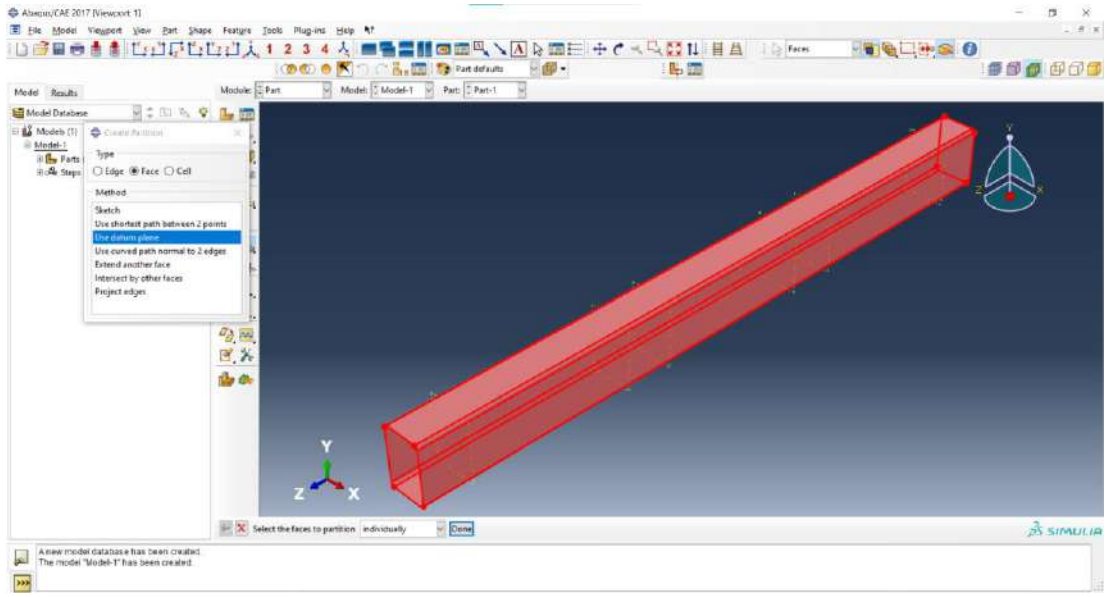


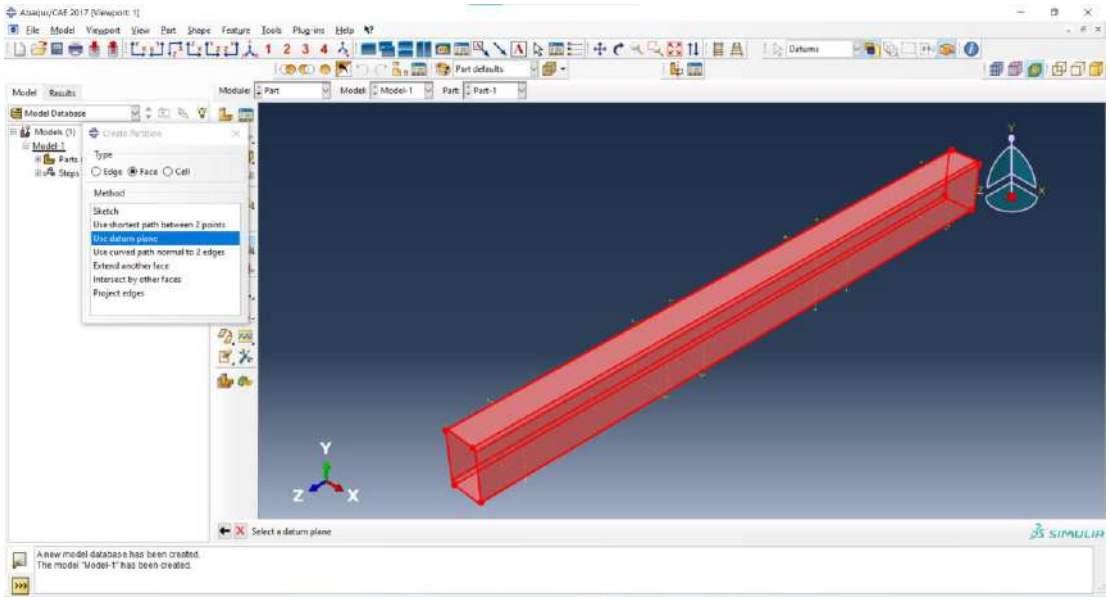


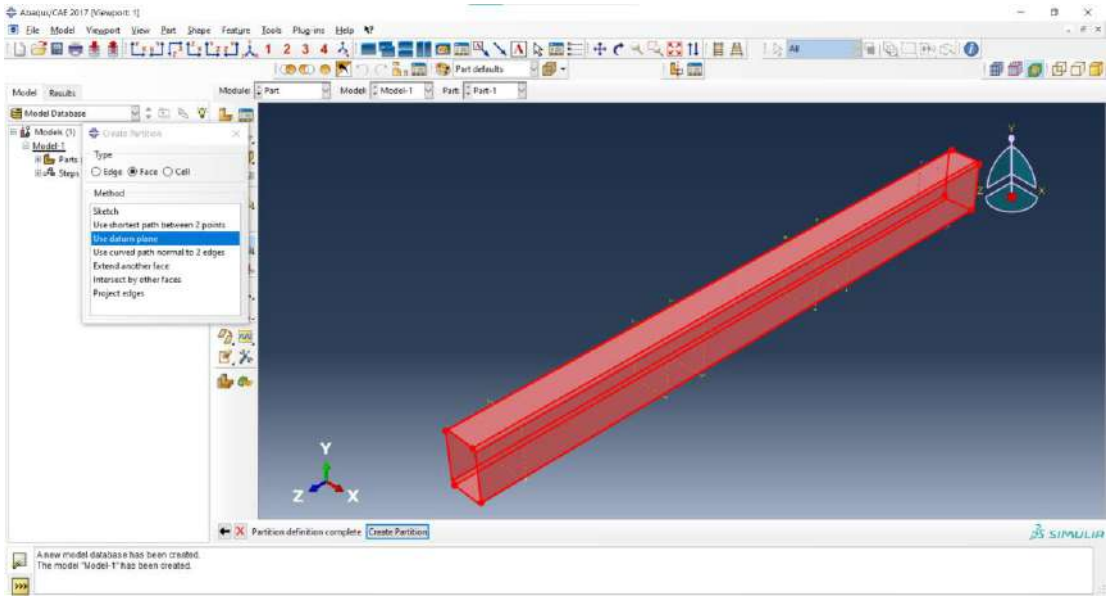
Setelah itu, Klik *Create Partition – Type Face – Use Datum Plane – Select* semua bagian model – Klik *Done* – Klik *Datum Plane* (Garis kuning putus-putus) yang sudah dibuat – Klik *Create Partition*. Lakukan secara berulang sampai semua *Datum Plane* sudah terpartisi.

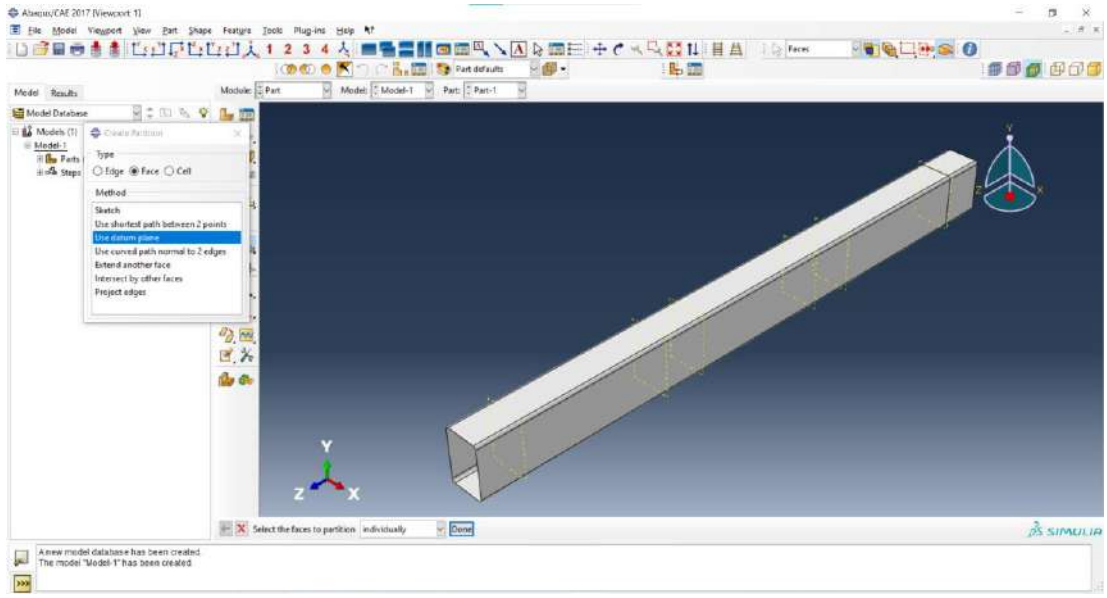


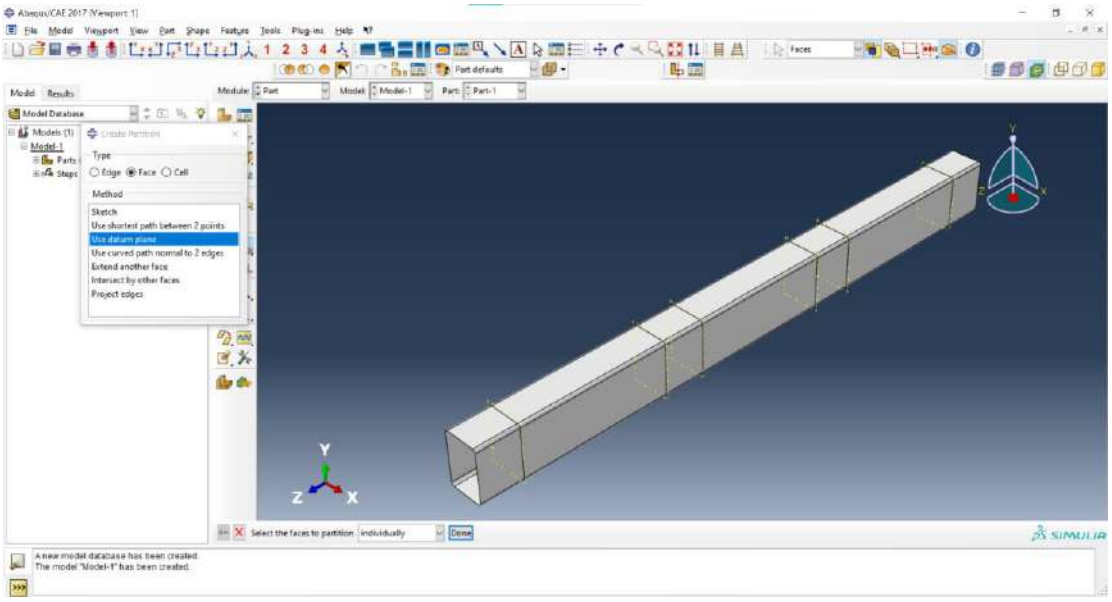



Select the faces to partition
individually
Done
A new model database has been created.
The model "Model-1" has been created.

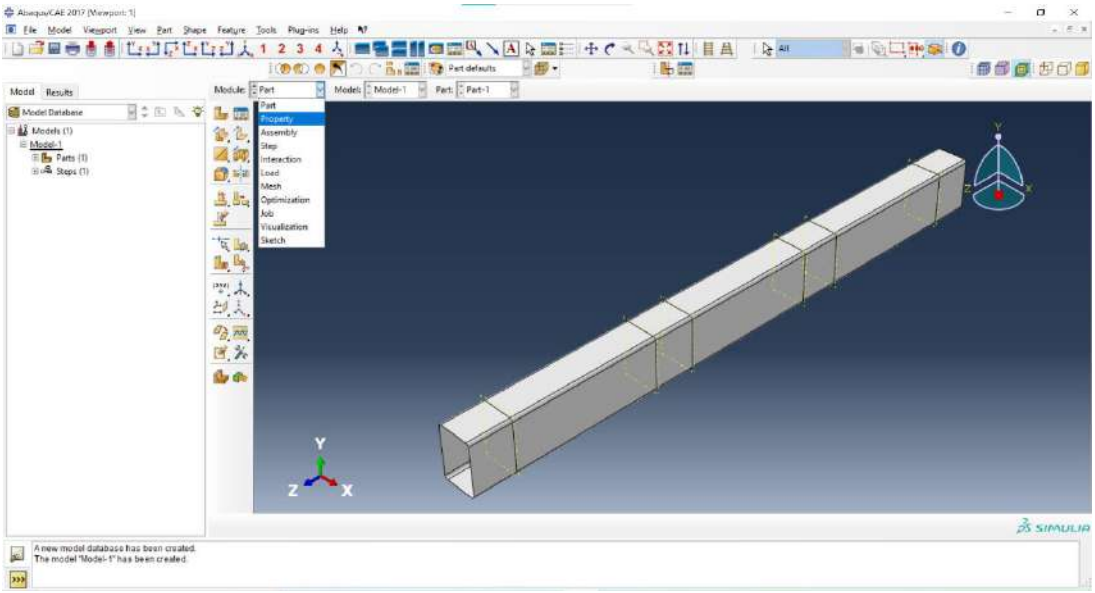

Select a datum plane
A new model database has been created.
The model "Model-1" has been created.


Partition definition complete
Create Partition
A new model database has been created.
The model "Model-1" has been created.

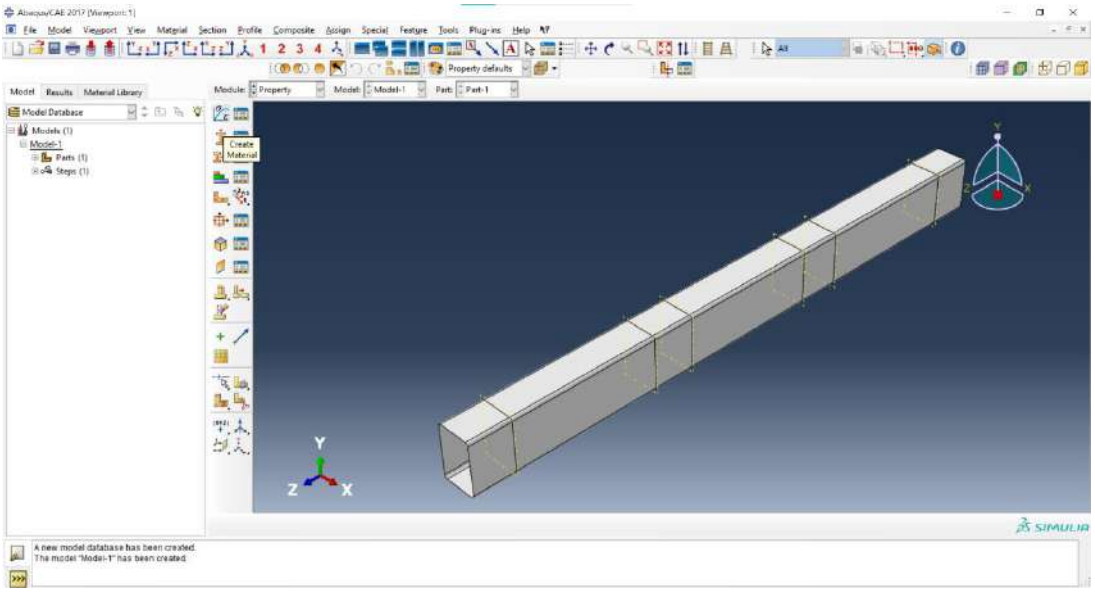

Abaqus/CAE 2017 [Viewport: 1]
File Model Viewport View Part Shape Feature Tools Plug-ins Help
Model Results
Model Database
Models (1)
Model-1
Parts
Steps
Module: Part
Model: Model-1
Part: Part-1
Create Partition
Type
Edge Face Cell
Method
Sketch
Use shortest path between 2 points
Use datum plane
Use curved path normal to 2 edges
Extend another face
Intersect by other faces
Project edges
Select the faces to partition
individually
Done
SIMULIA
A new model database has been created.
The model "Model-1" has been created.

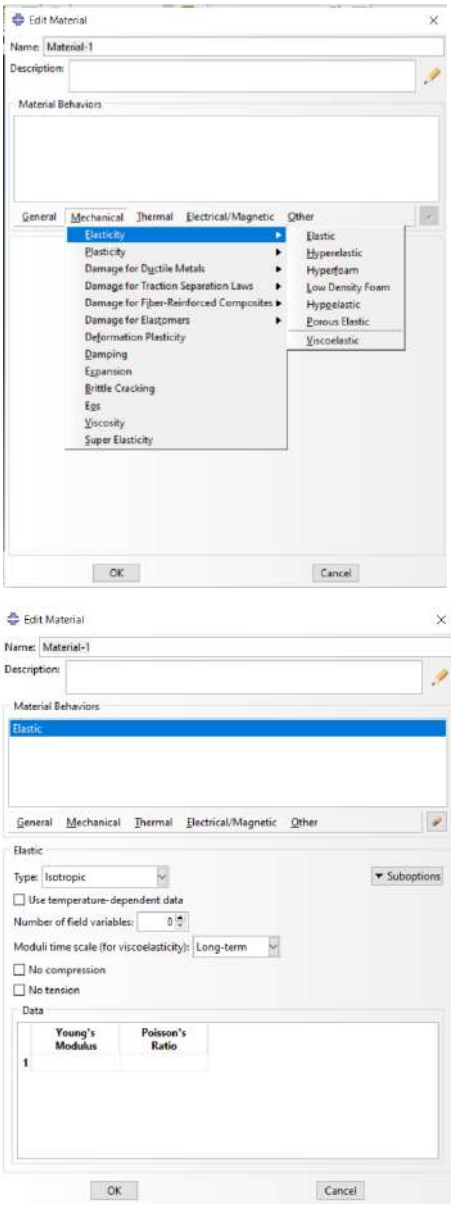

Abaqus/CAE 2017 [Viewport: 1]
File Model Viewport View Part Shape Feature Tools Plug-ins Help
Model Results
Model Database
Models (1)
Model-1
Parts
Steps
Module: Part
Model: Model-1
Part: Part-1
Create Partition
Type
Edge Face Cell
Method
Sketch
Use shortest path between 2 points
Use datum plane
Use curved path normal to 2 edges
Extend another face
Intersect by other faces
Project edges
Select the faces to partition
individually
Done
SIMULIA
A new model database has been created.
The model "Model-1" has been created.

3. Membuat Properti Material, pada *Module:* Pilih *Property*.

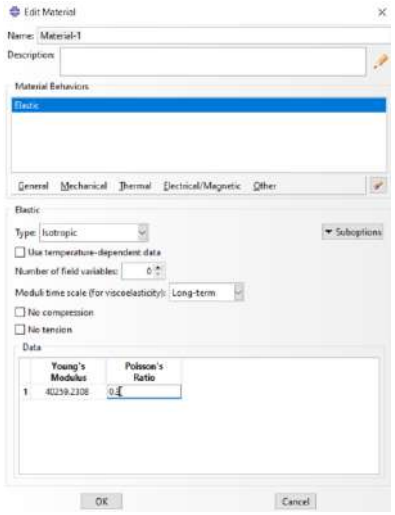


Klik *Create Material* – Pada *Edit Material* – Klik *Mechanical – Elasticity – Elastic*.

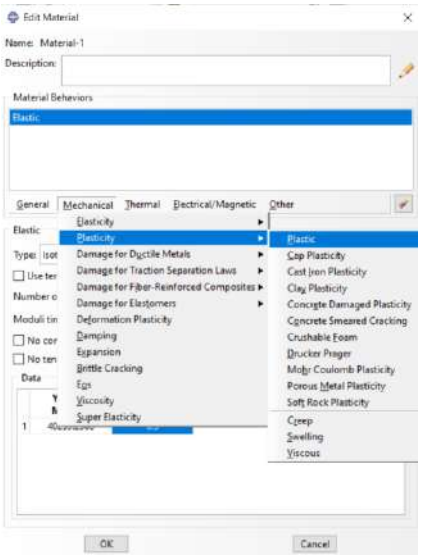


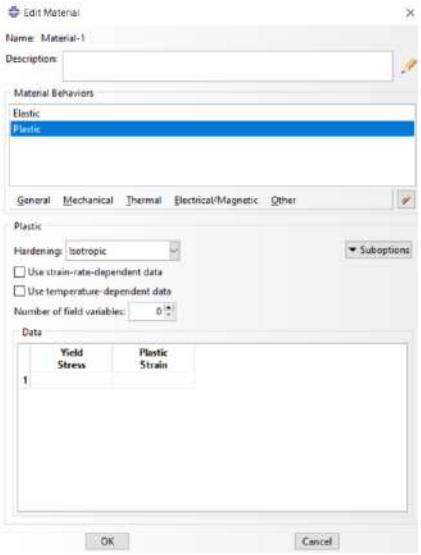


Input nilai *Young's Modulus* yaitu 40259.2308 MPa dan *Poisson's Ratio* yaitu 0.3.



Pada *Edit Material* – Klik *Mechanical – Plasticity – Plastic*.

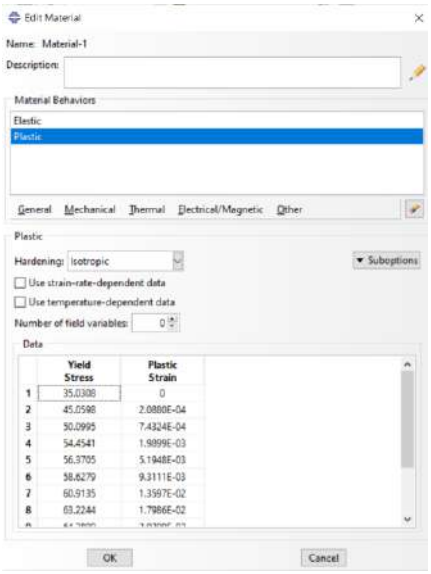




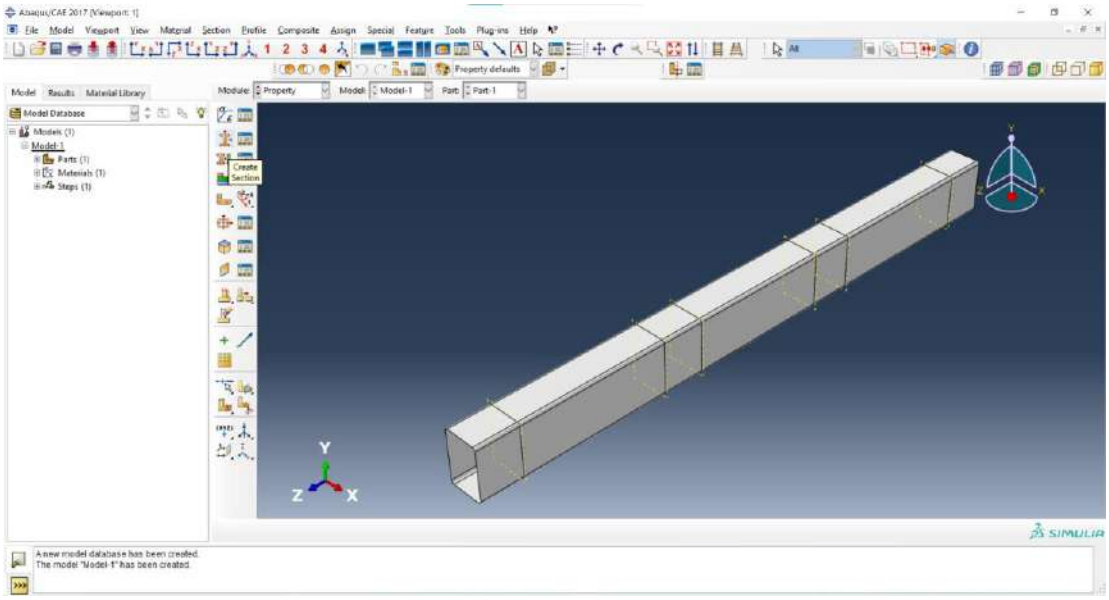
Input nilai *plastic* sesuai dengan tabel – Klik Ok.

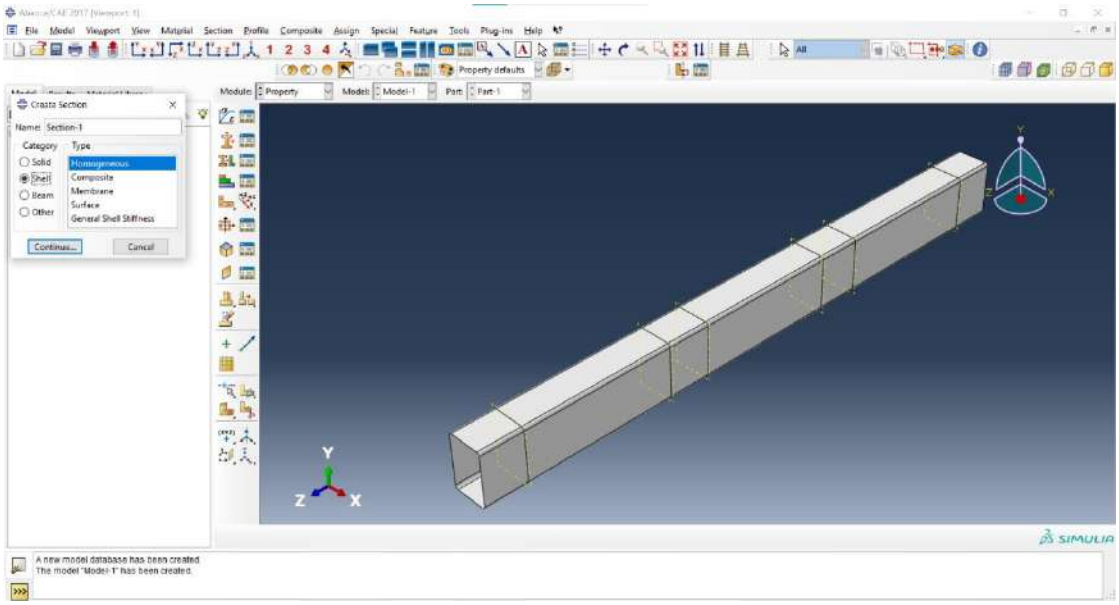
σ_{true} (MPa)	ε_{true}
35.0308	0
45.0598	2.0880E-04
50.0995	7.4324E-04
54.4541	1.9899E-03
56.3705	5.1948E-03
58.6279	9.3111E-03
60.9135	1.3597E-02
63.2244	1.7986E-02
64.3890	2.0209E-02
65.5596	2.2448E-02
66.7360	2.4700E-02
67.9183	2.6964E-02

68.9720	2.8981E-02

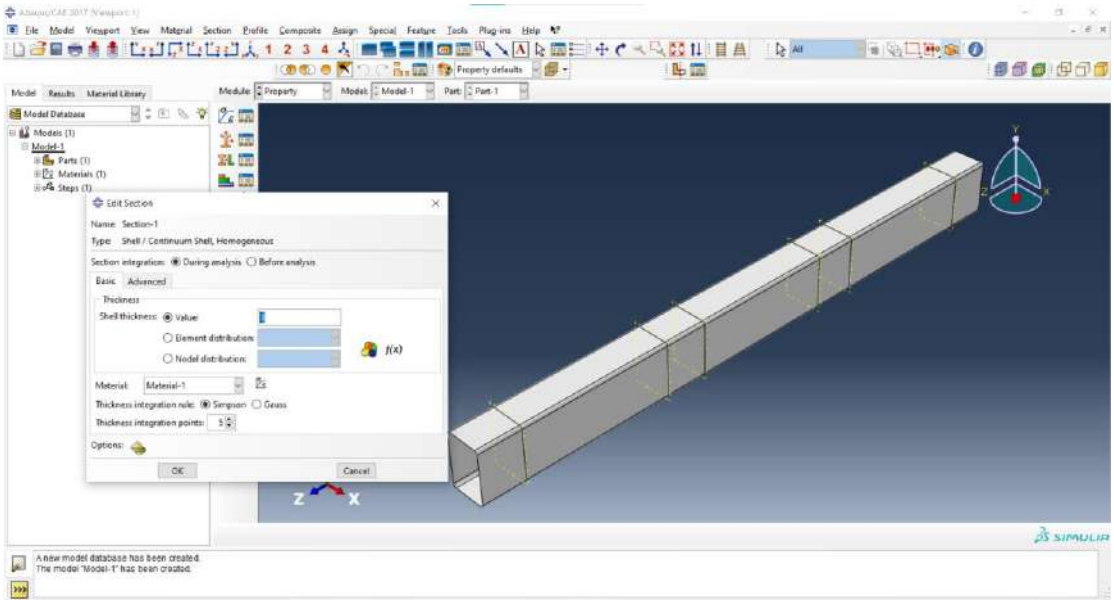


Membuat *Section Properties*, Klik *Create Section* – *Category Shell* – *Type Homogeneous* – Klik *Continue*…

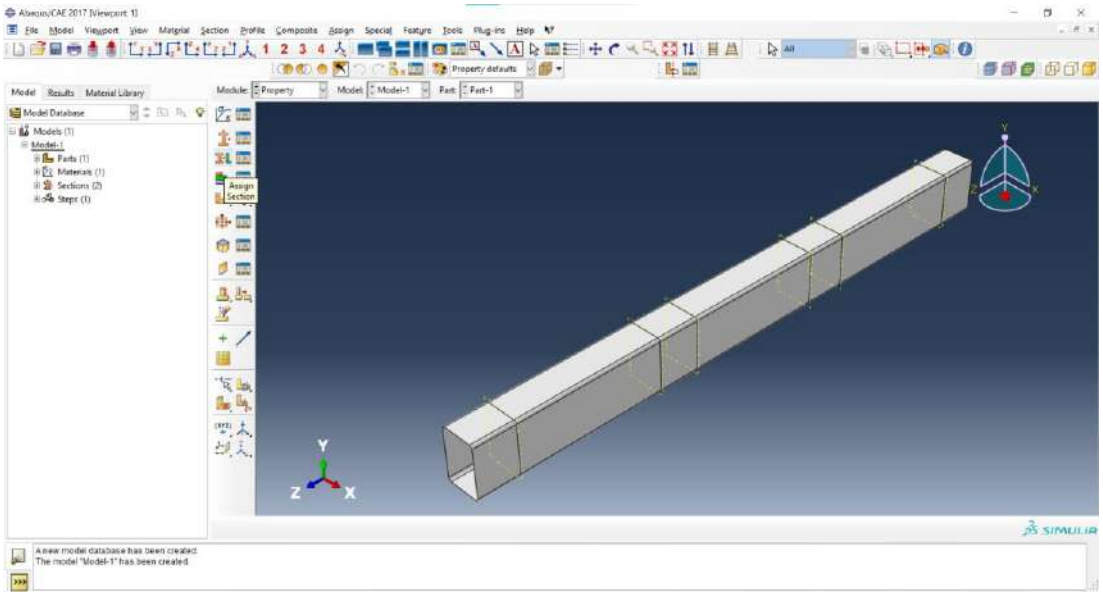


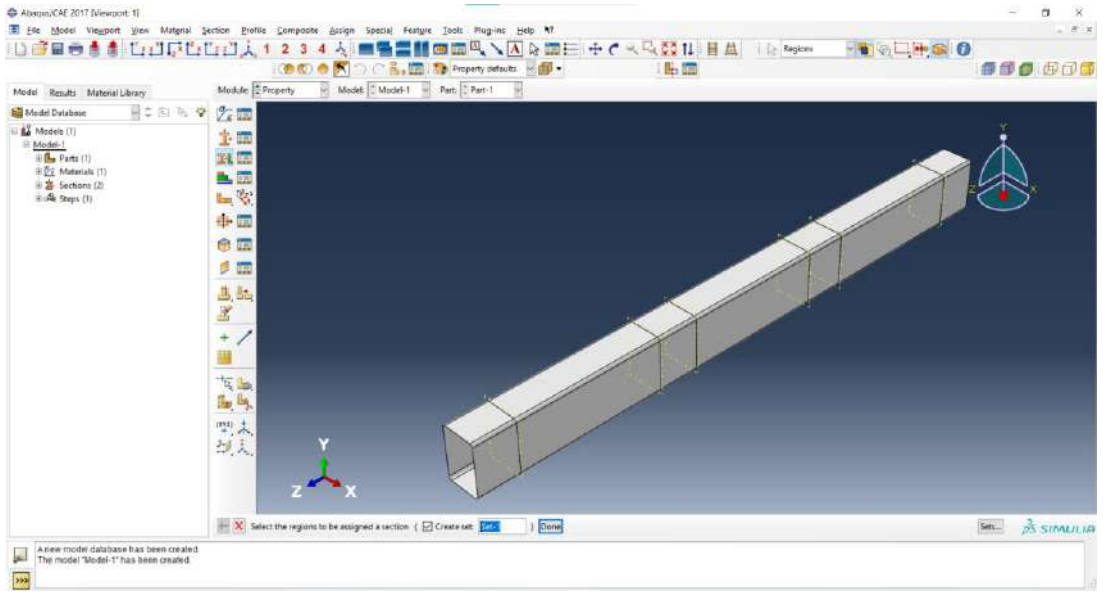


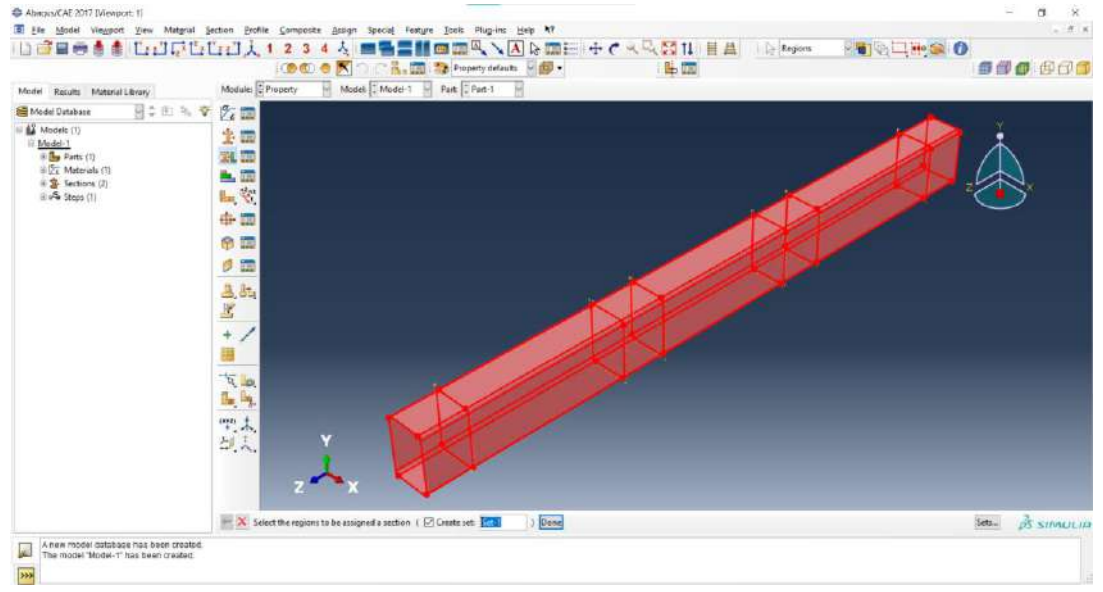
Pada *Edit Section – Shell Thickness: Value* diisi sesuai tebal profil yaitu 3 – Klik Ok. Lalu ulangi untuk *section* 2 dengan *Shell Thickness: Value* diisi 12 untuk penebalan pada bagian tumpuan dan *load*.

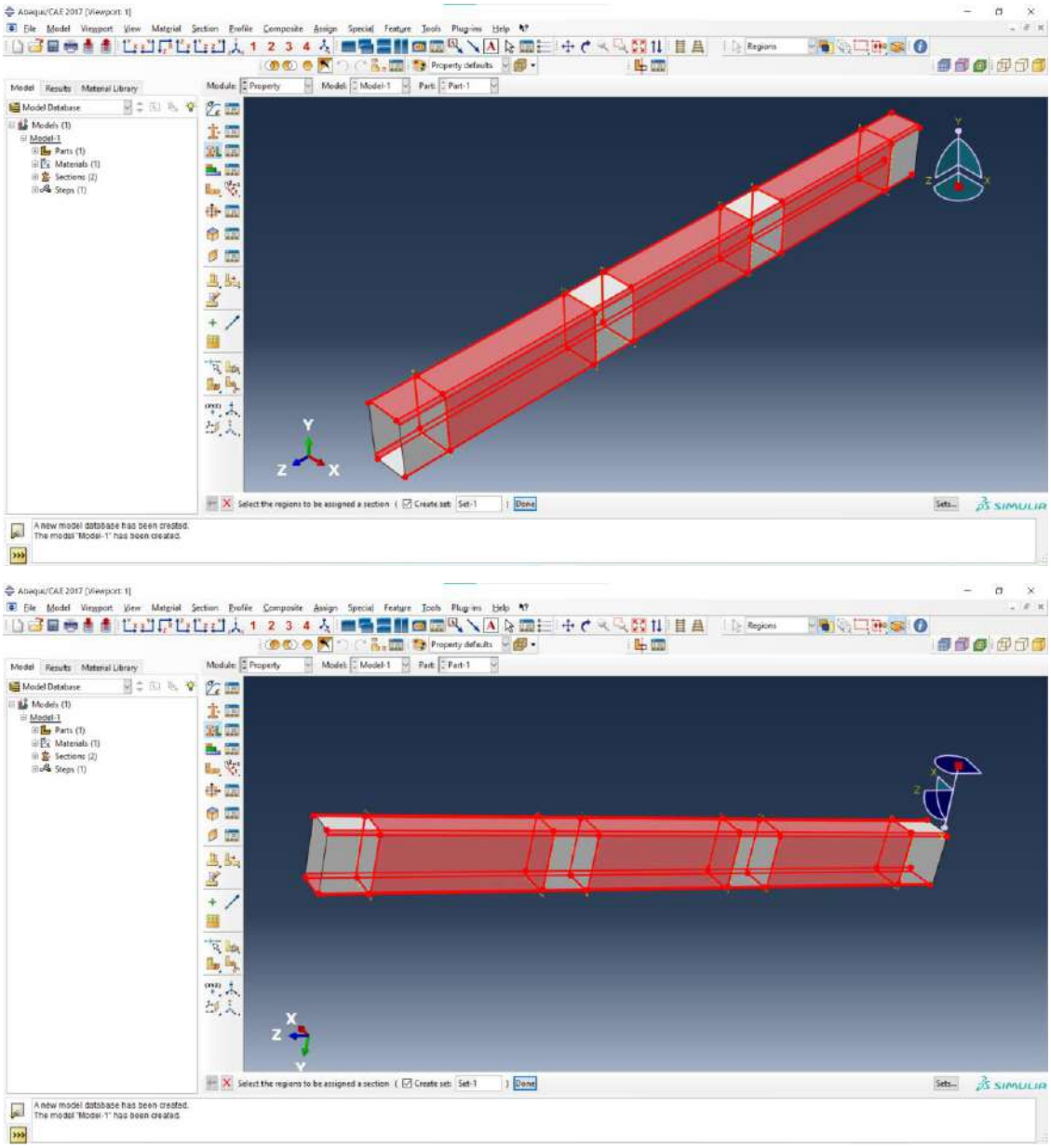


Setelah dibuat 2 *section*, Klik *Assign Section – Select* bagian-bagian yang tidak mengalami penebalan – Klik Done. Cara *select* bagian-bagian yang tidak mengalami penebalan dengan cara *select* semua bagian model lalu tahan tombol Ctrl dan klik bagian-bagian yang mengalami penebalan. Untuk memutar model, dapat menekan tombol Ctrl, Alt dan bagian kiri *mouse*.

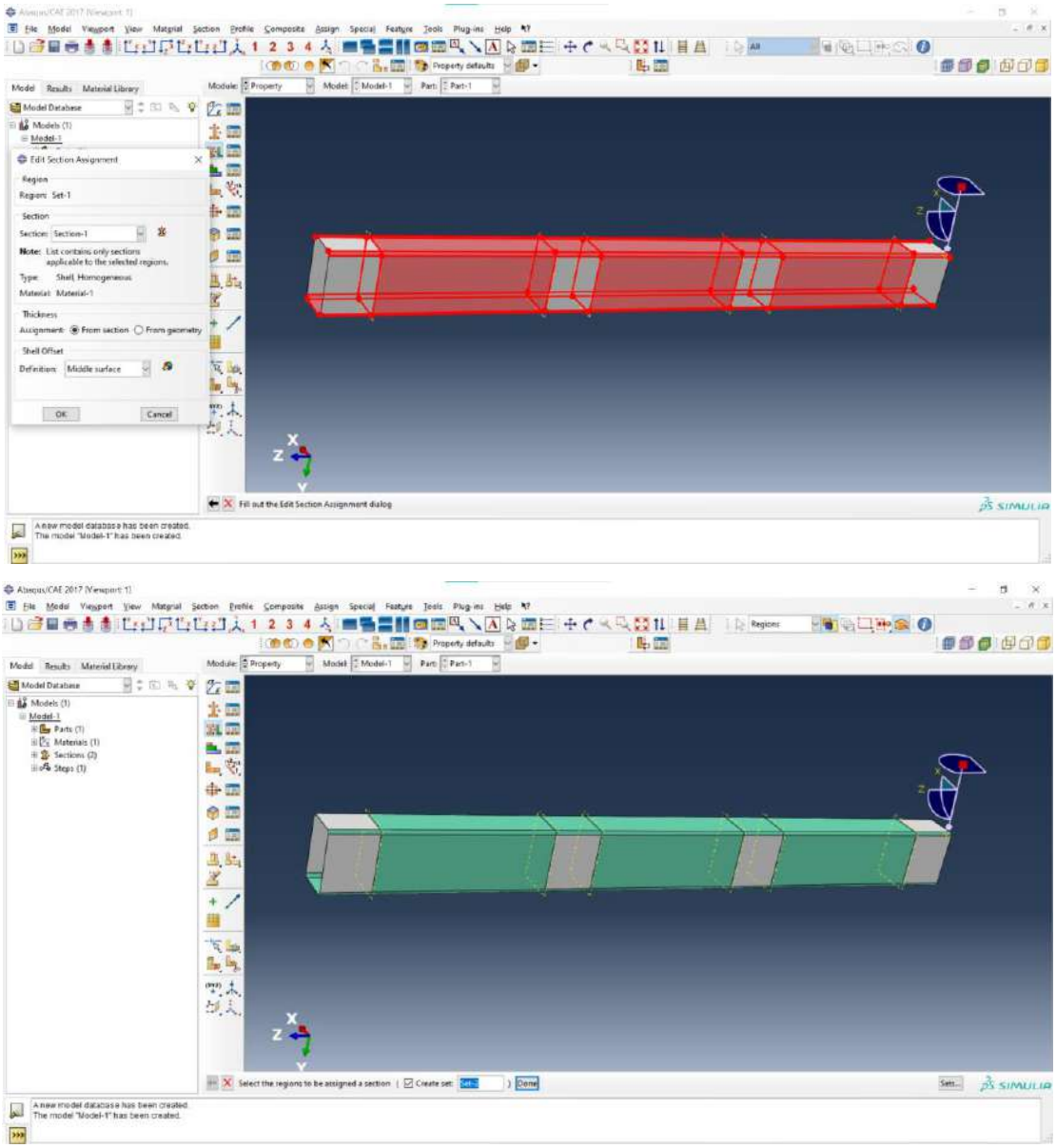




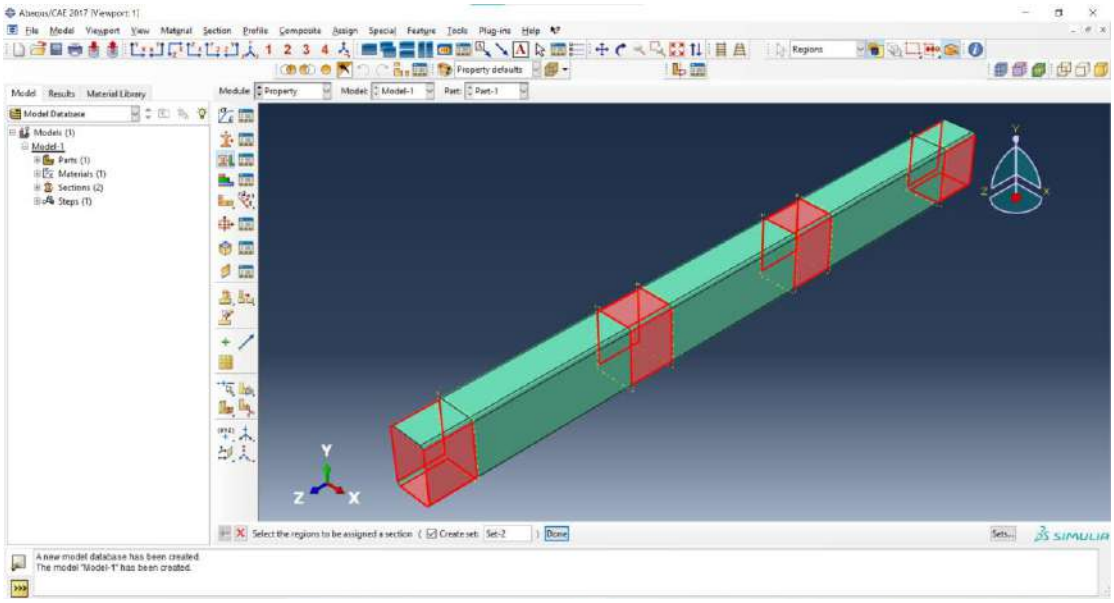




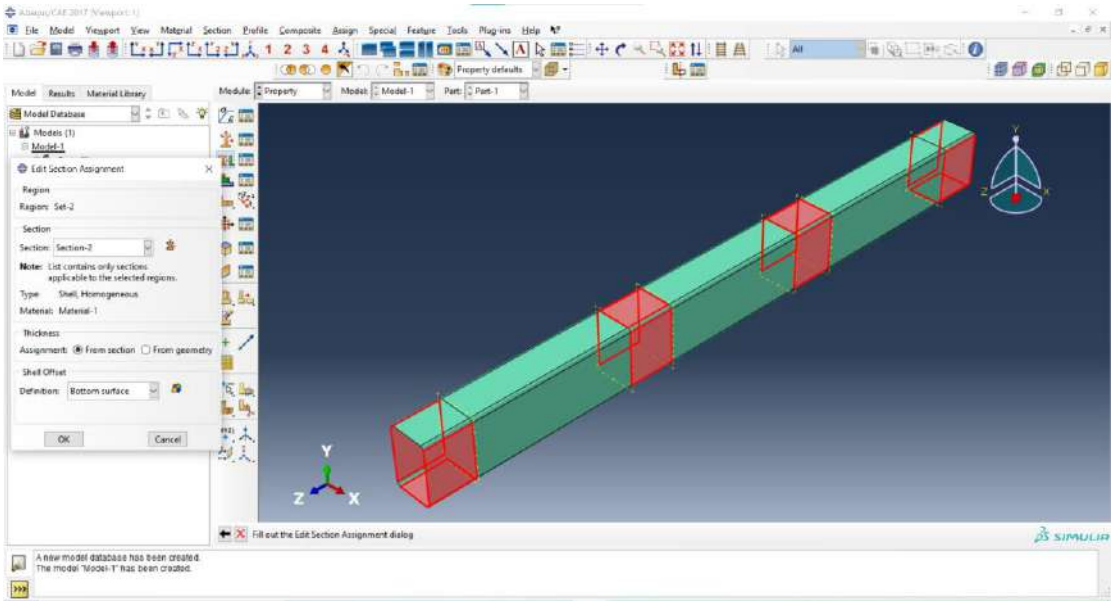
Pada *Edit Section Assignment*, gunakan Section-1 dengan *Shell Offset – Definition: Middle Surface* – Klik Ok.

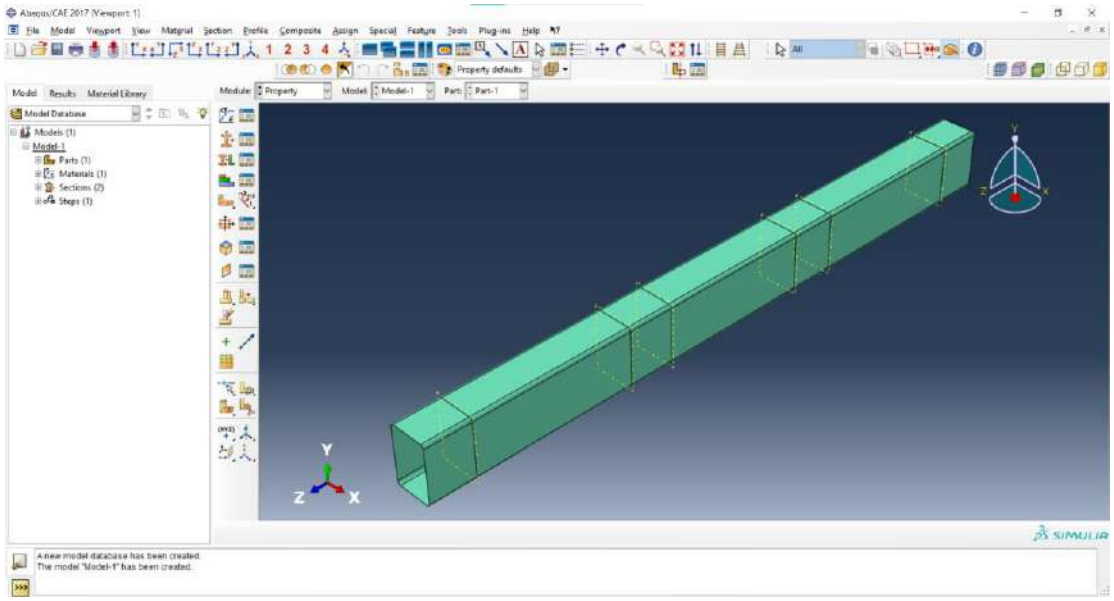


Lakukan juga untuk bagian-bagian yang mengalami penebalan dengan *Assign Section.* Cara *select* dengan menahan tombol *Shift* dan bagian kiri *mouse*.

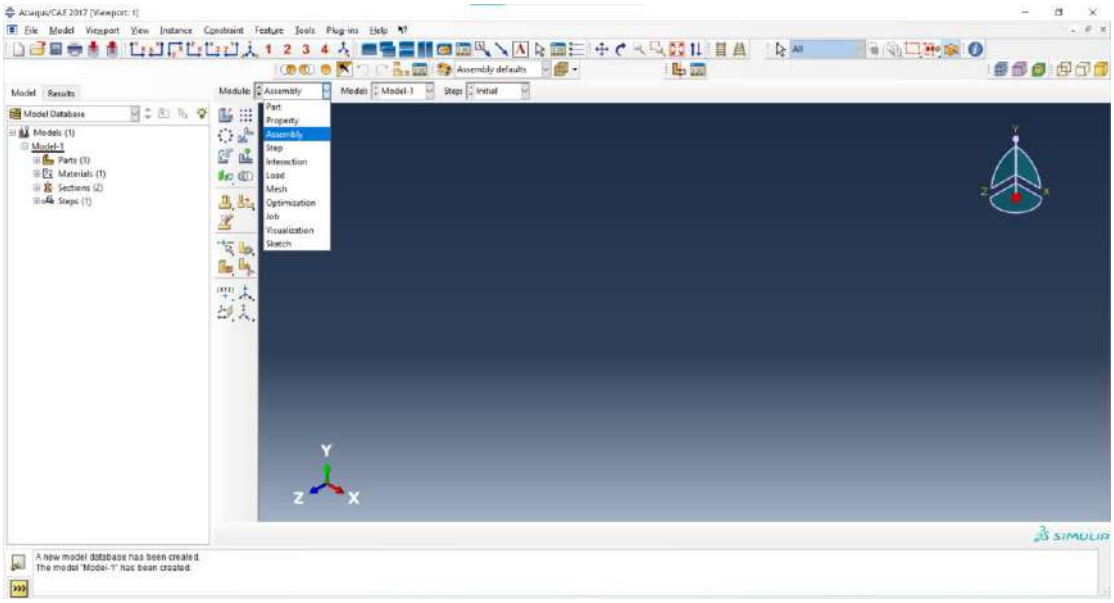


Pada *Edit Section Assignment*, gunakan Section-2 dengan *Shell Offset – Definition: Bottom Surface* – Klik Ok.

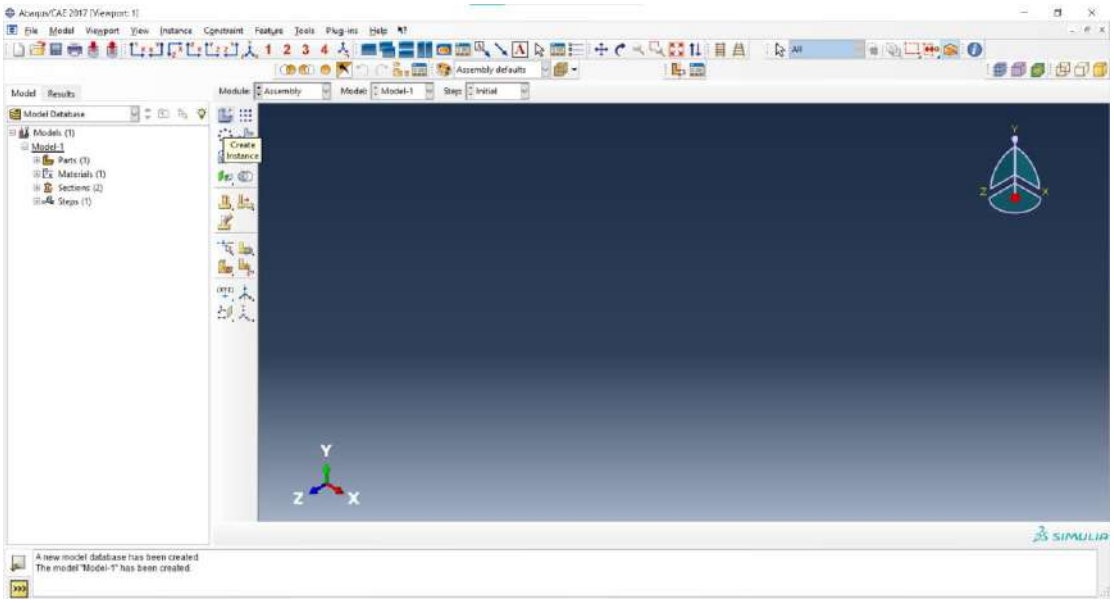


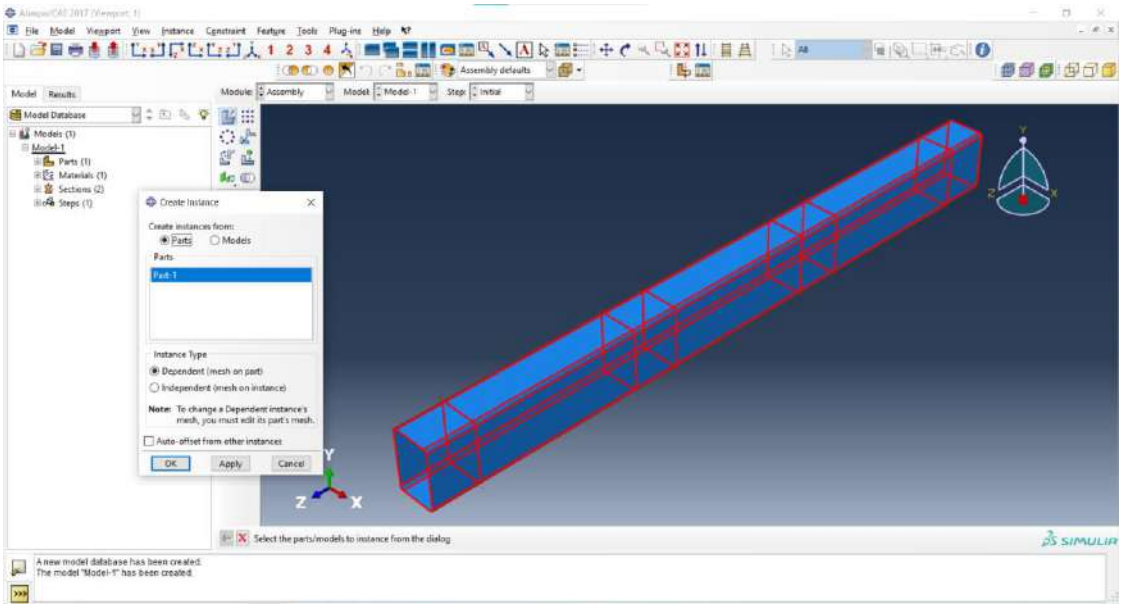


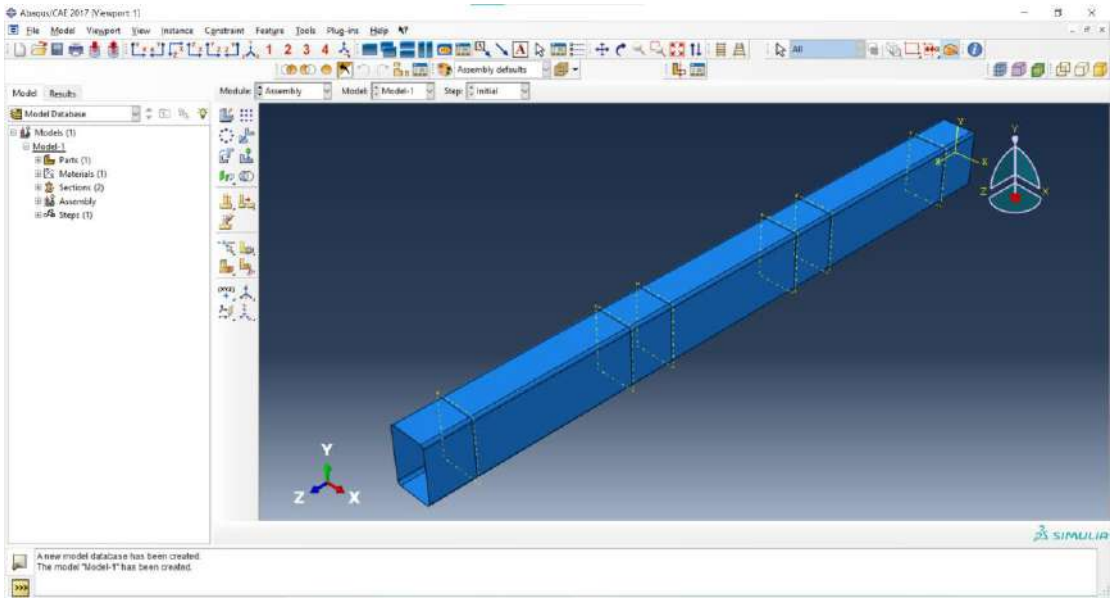
4. Memasukan *Part*, pada *Module:* Pilih *Assembly*.



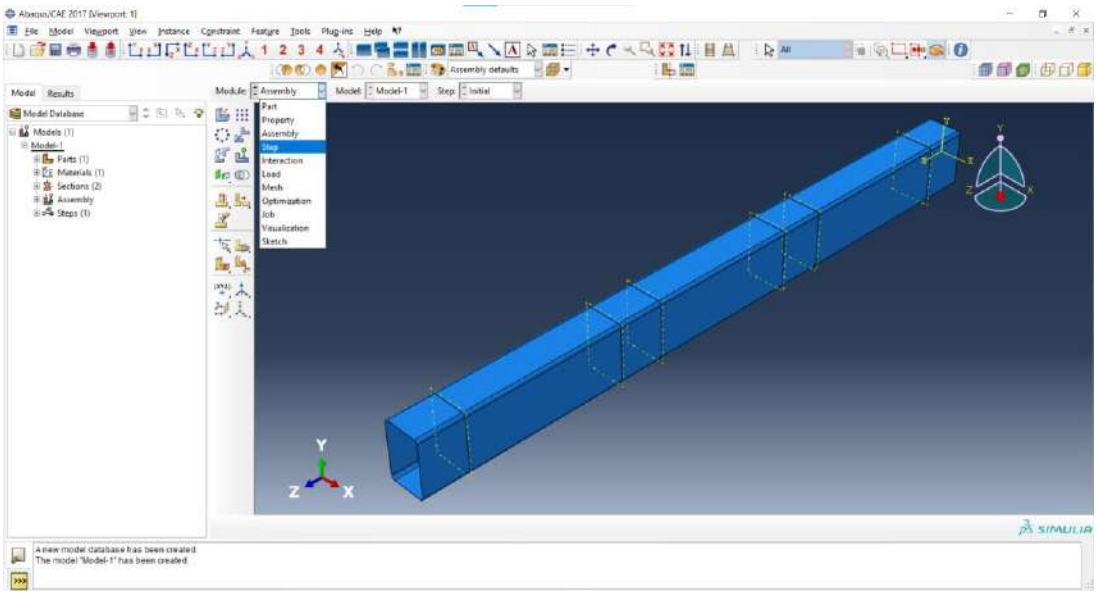
Klik *Create Instance* – *Create Instance From: Parts* – Klik Part-1 – *Instance Type Dependent* (*mesh on part*) – Klik Ok.



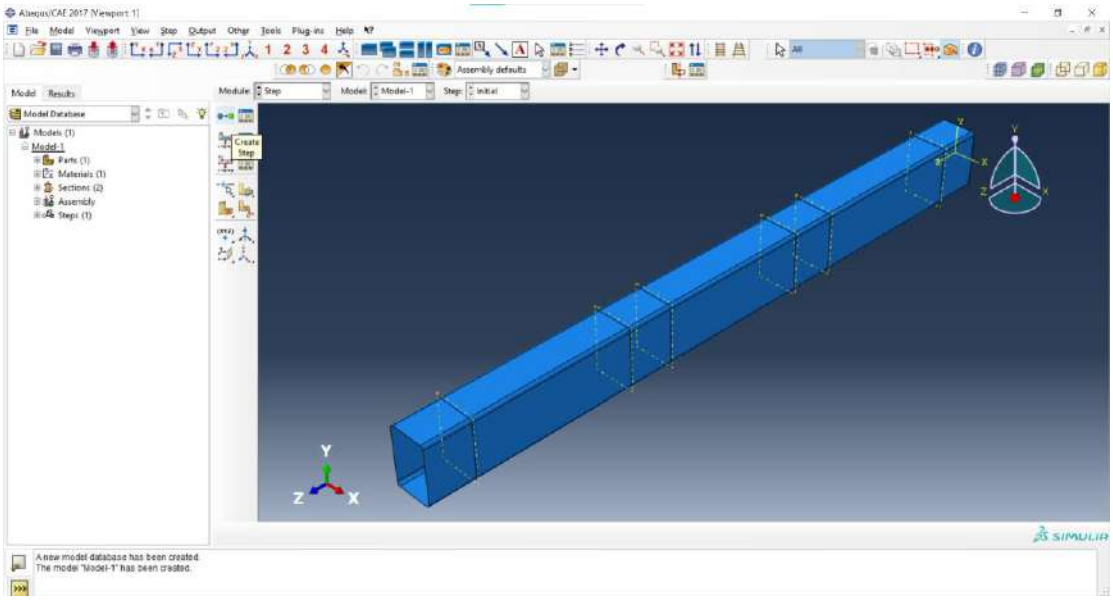

Create Instance
Create instances from:
Parts
Models
Parts
Part-1
Instance Type
Dependent (mesh on part)
Independent (mesh on instance)
Note: To change a Dependent instance's mesh, you must edit its part's mesh.
Auto-offset from other instances
OK
Apply
Cancel
Select the parts/models to instance from the dialog

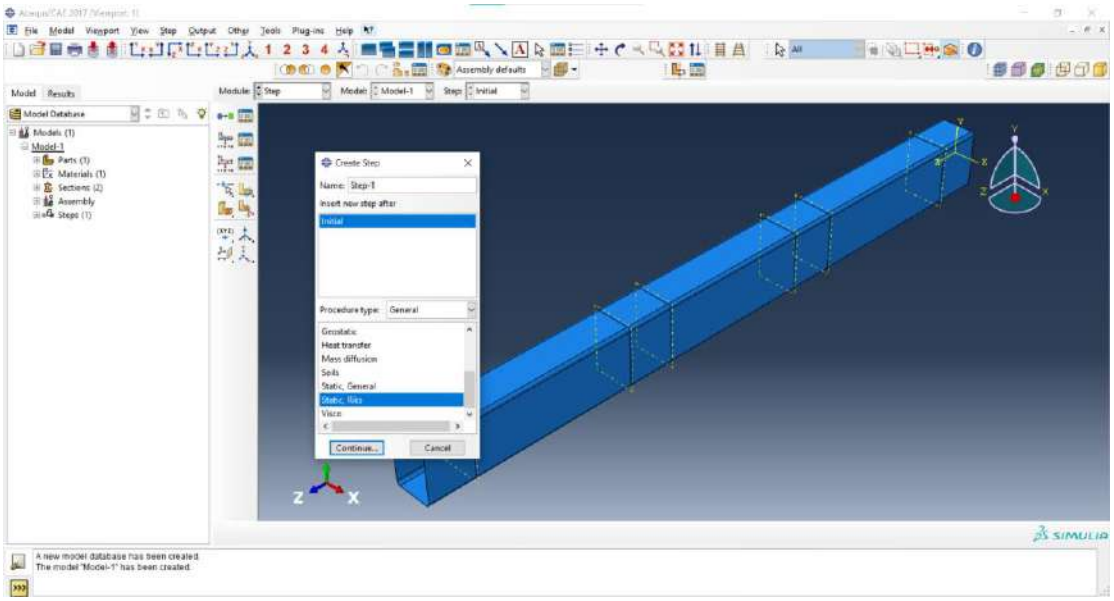


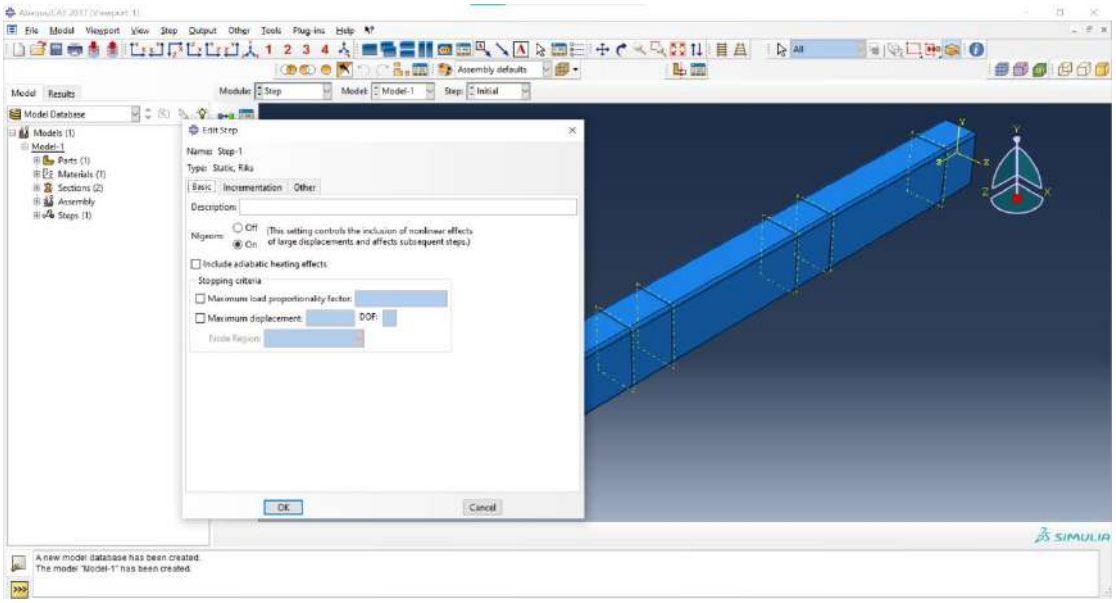
5. Membuat *Step*, pada *Module:* Pilih *Step*.

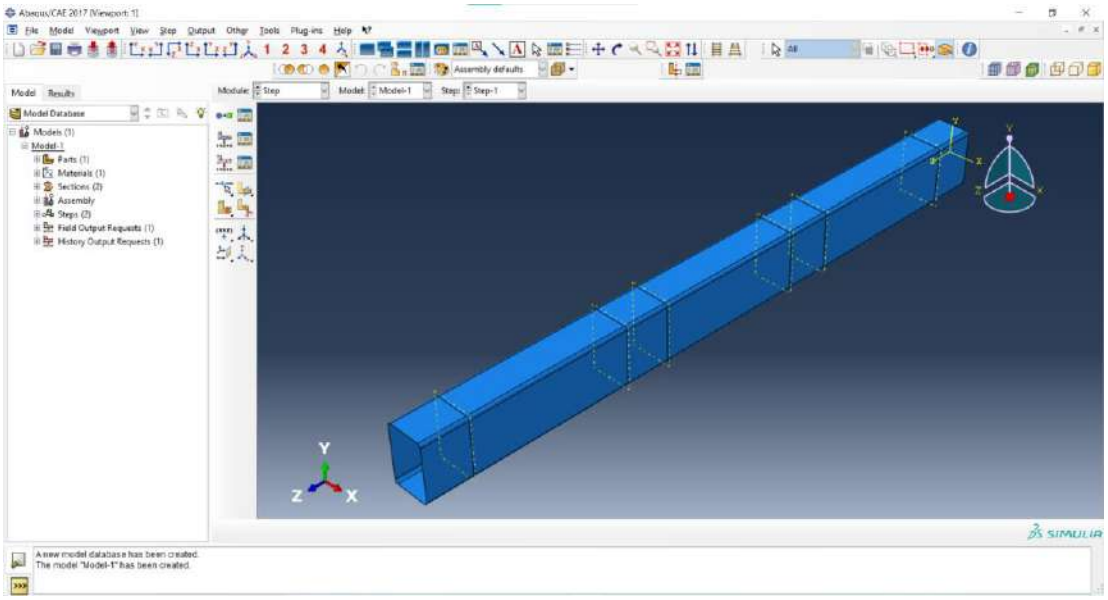


Create Step – Static, Riks – Continue… – Pada *Edit Step, NLGEOM* **ON** – Klik Ok.

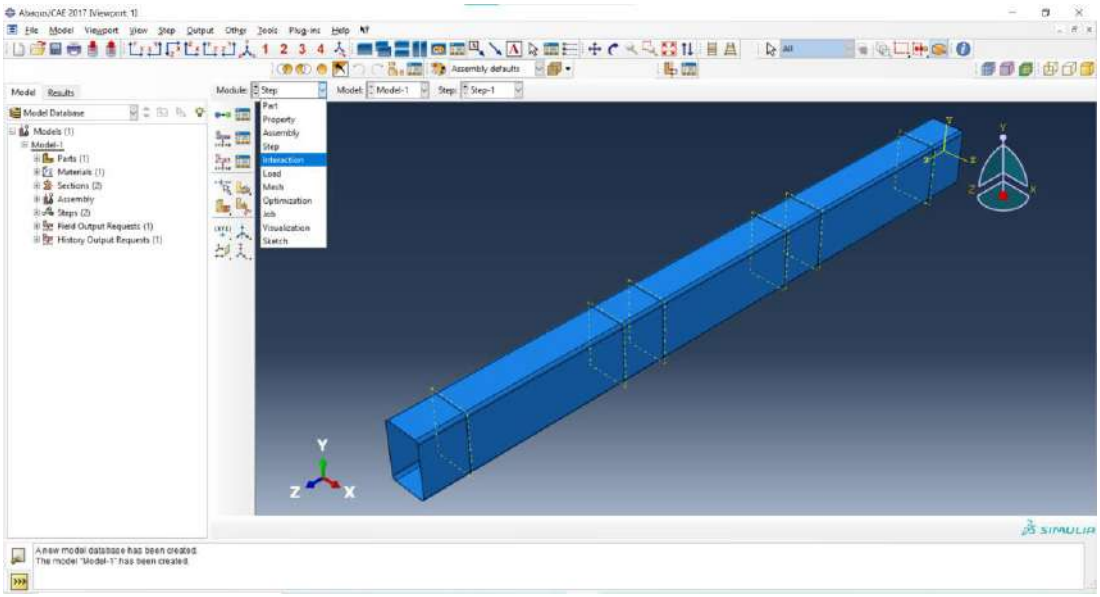



Create Step
Name: Step-1
Insert new step after
Initial
Procedure type: General
Static, General
Static, Riks
Continue...
Cancel
A new model database has been created.
The model "Model-1" has been created.


Edit Step
Name: Step-1
Type: Static, Riks
Basic
Incrementation
Other
Description:
Nlgeom:
Off
On
(This setting controls the inclusion of nonlinear effects of large displacements and affects subsequent steps.)
Include adiabatic heating effects
Stopping criteria
Maximum load proportionality factor:
Maximum displacement:
DOF:
Node Region:
OK
Cancel
A new model database has been created.
The model "Model-1" has been created.

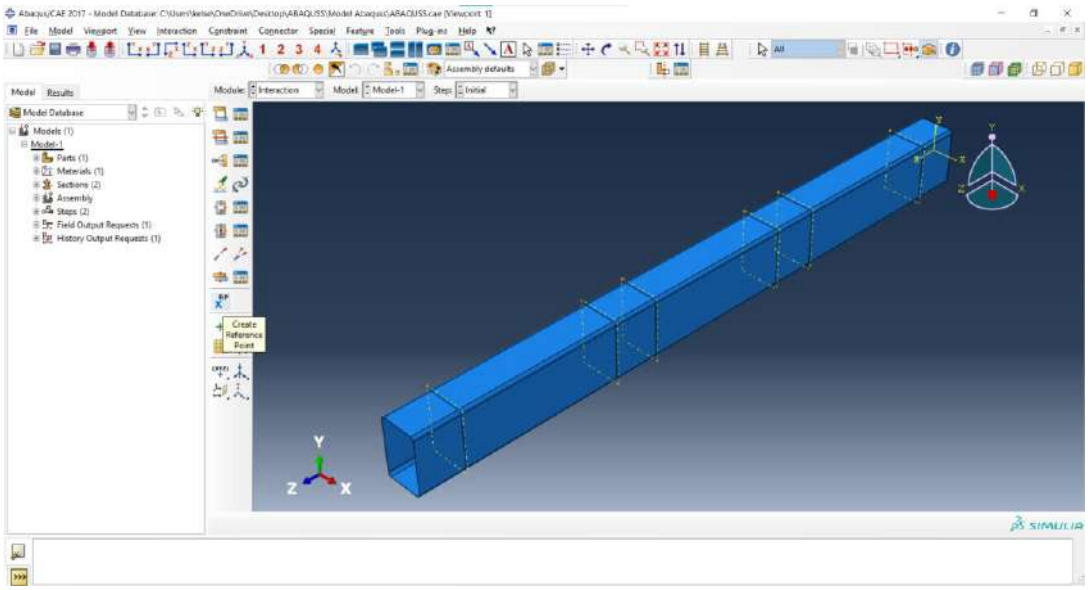

Field Output Requests (1)
History Output Requests (1)
A new model database has been created.
The model "Model-1" has been created.

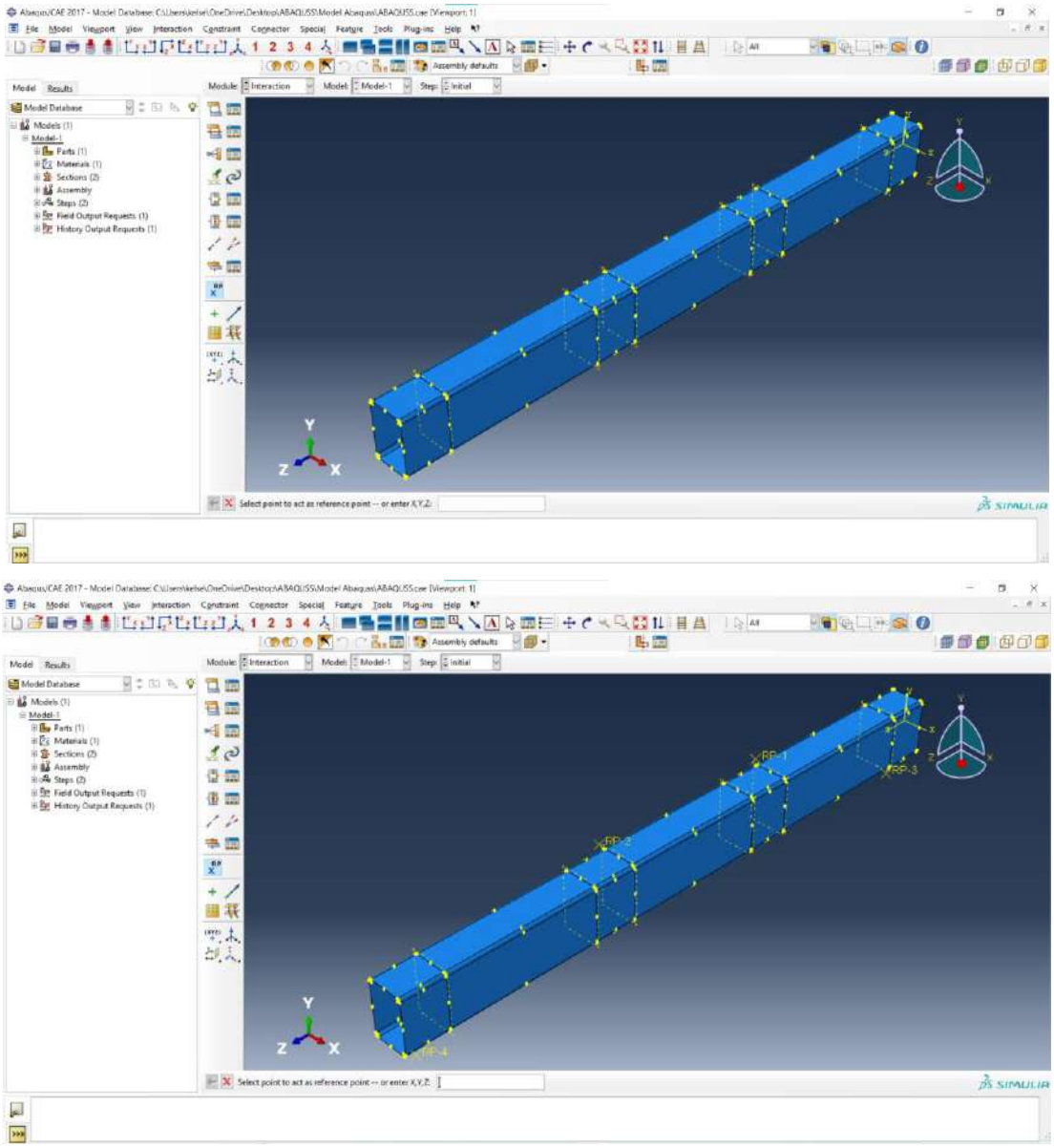
6. Membuat *Interaction*, pada *Module:* Pilih *Interaction*.



Klik *Create Reference Point* – Input Titik Tengah dari *section* yang mengalami penebalan seperti pada tabel.

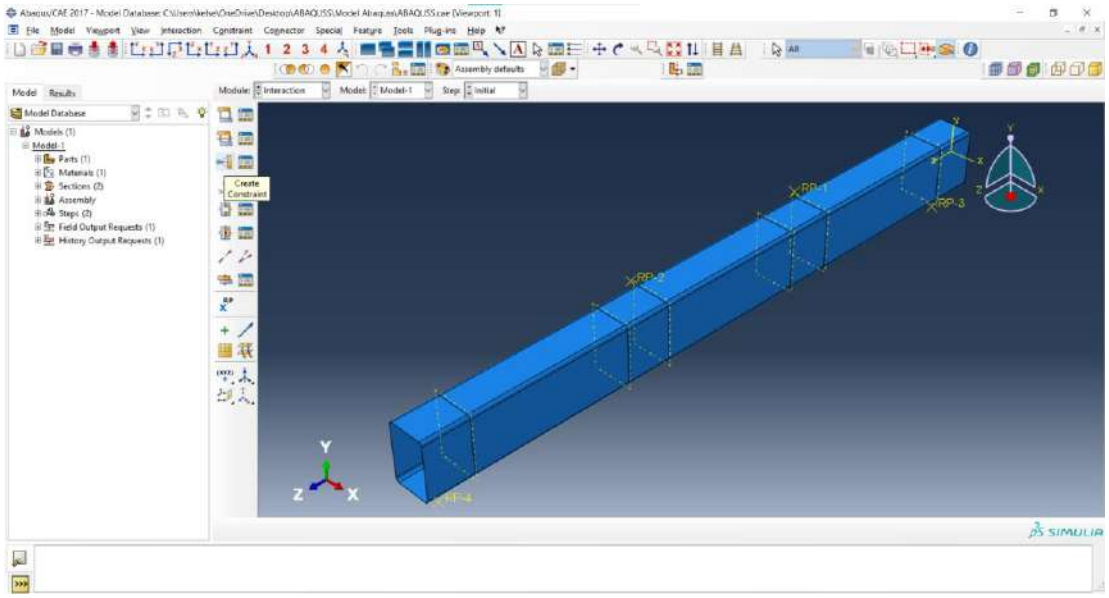
	X	Y	Z
RP1	0	105	455
RP2	0	105	845
RP3	0	-105	45
RP4	0	-105	1255

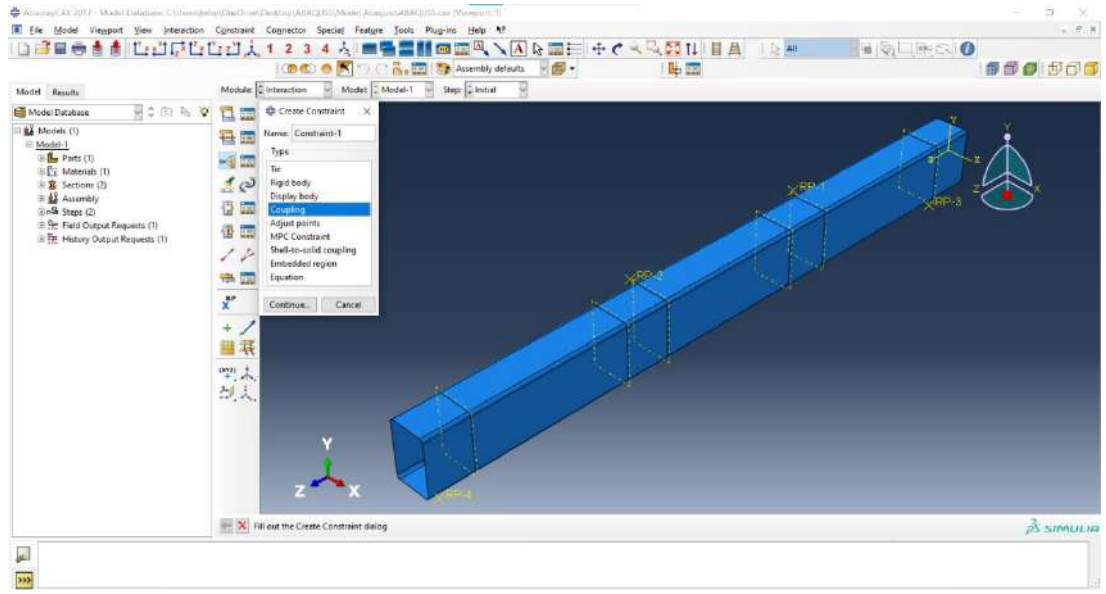


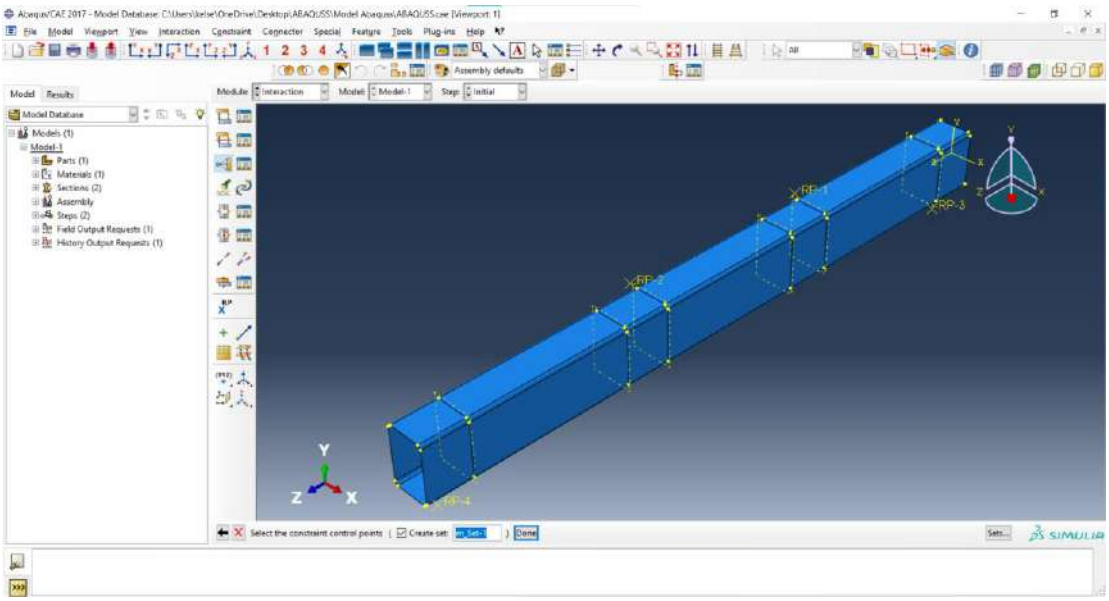


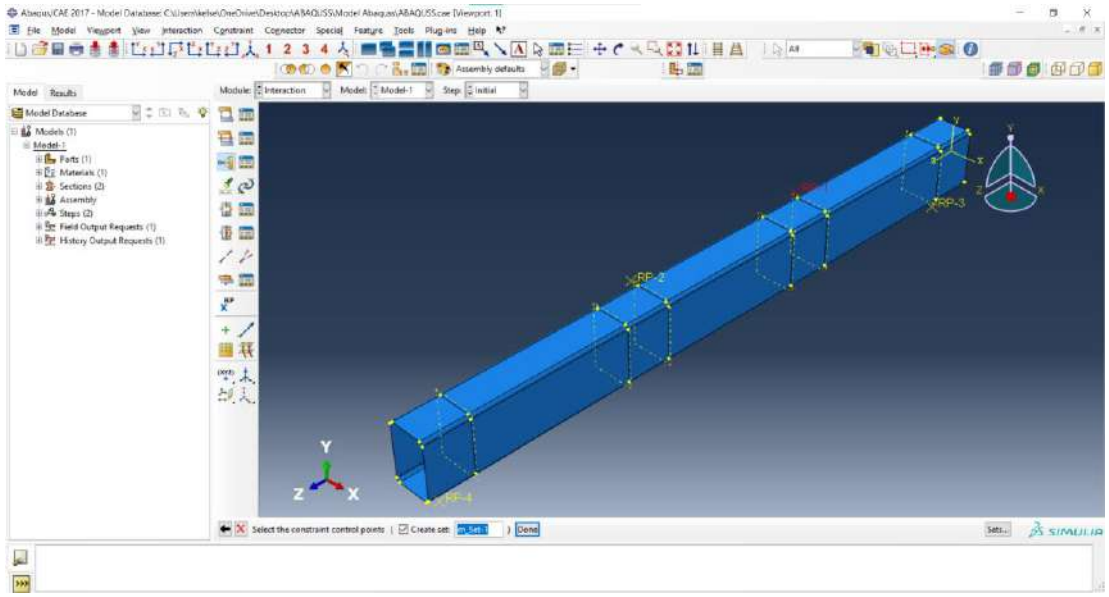
Setelah semua *reference point* (RP) sudah diinput, klik X pada "*Select point to act as reference point – or enter* X,Y,*Z*:".

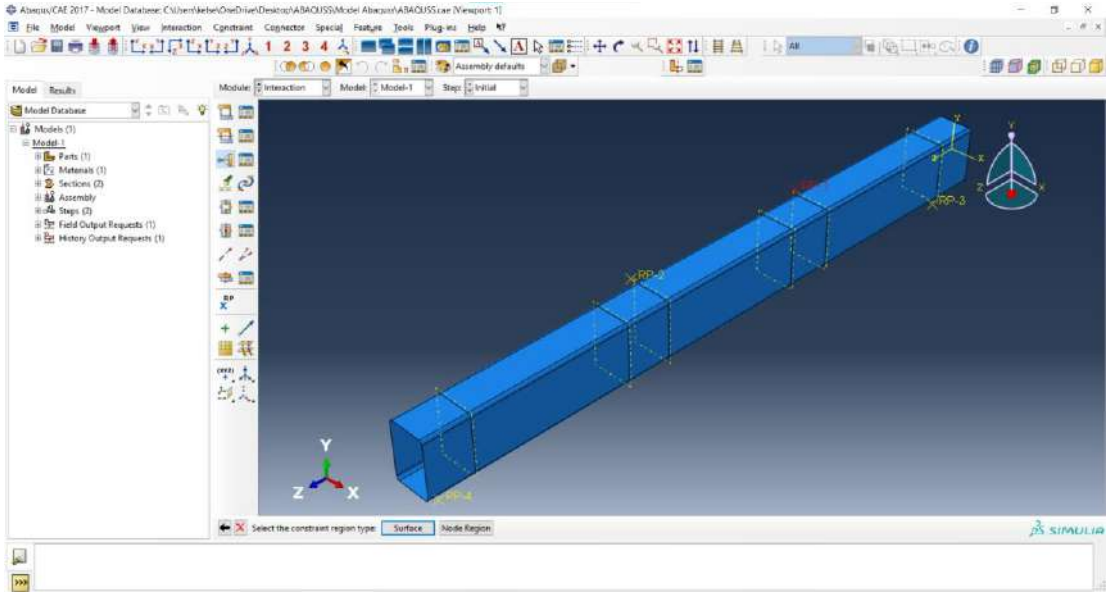
Klik *Create Constraint – Type Coupling – Continue…* – Klik RP-1 – Klik *Done* – Klik *Surface* – Klik Bidang di bawah RP-1 – Klik *Done* – Klik *Brown* – Pada *Edit Constraint, Coupling Type: Kinematic* – Klik Ok.

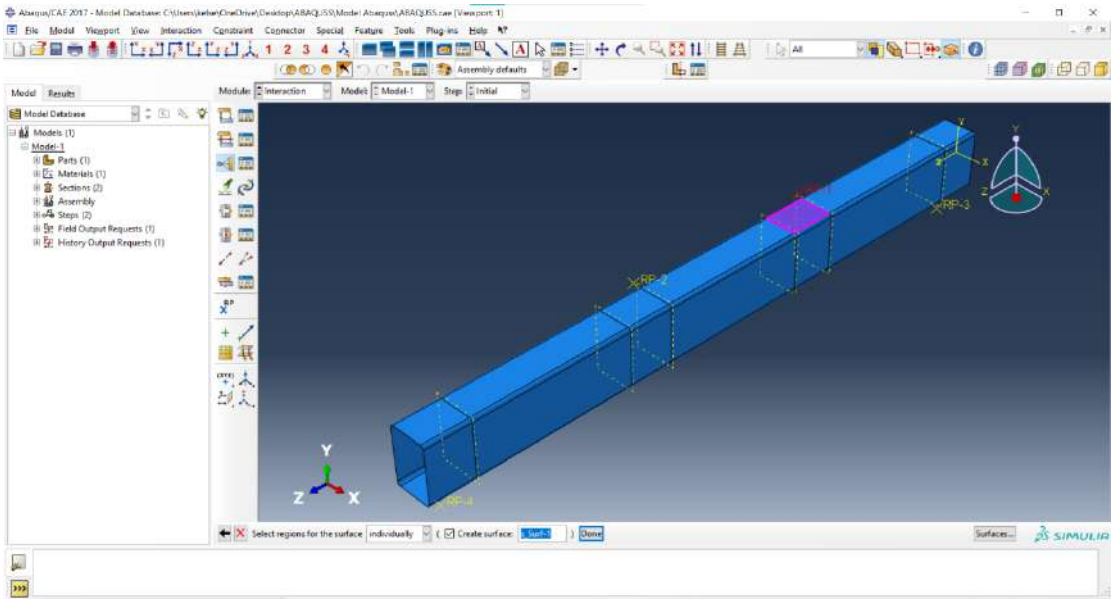

RP-1
RP-2
RP-3
RP-4
Y
Z
X

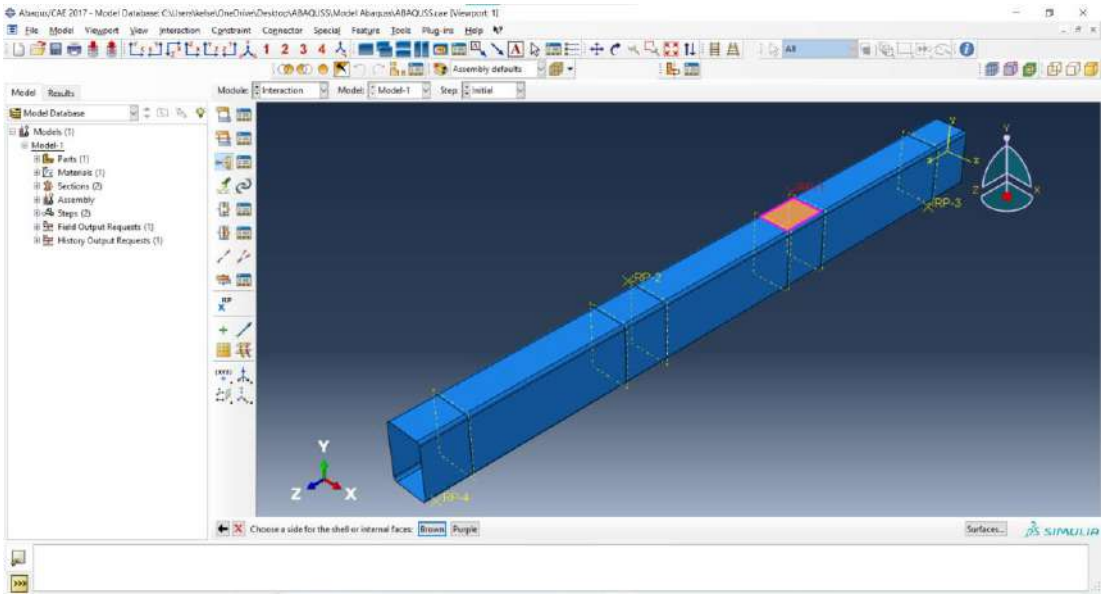

Create Constraint
Name: Constraint-1
Type
Tie
Rigid body
Display body
Coupling
Adjust points
MPC Constraint
Shell-to-solid coupling
Embedded region
Equation
Continue...
Cancel
Fill out the Create Constraint dialog
RP-1
RP-2
RP-3
RP-4

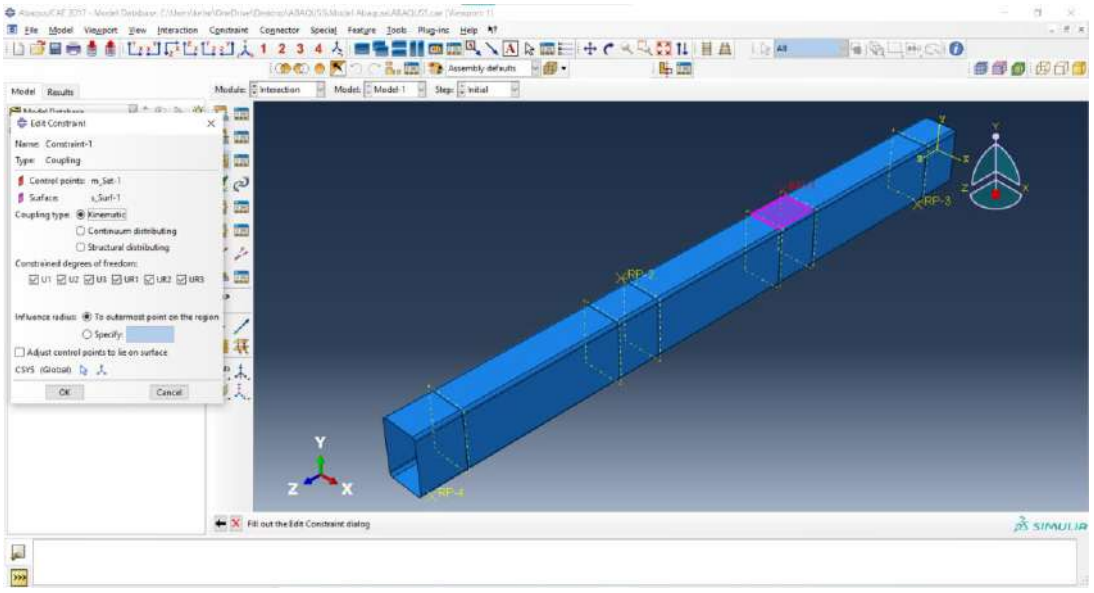

RP-1
RP-2
RP-3
RP-4
Y
Z
X

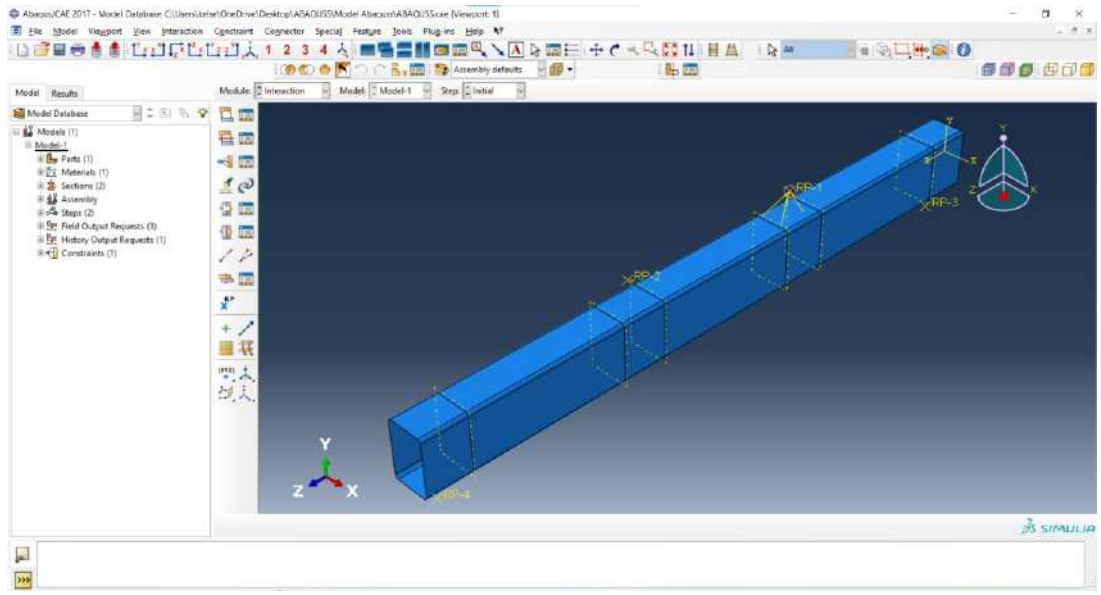




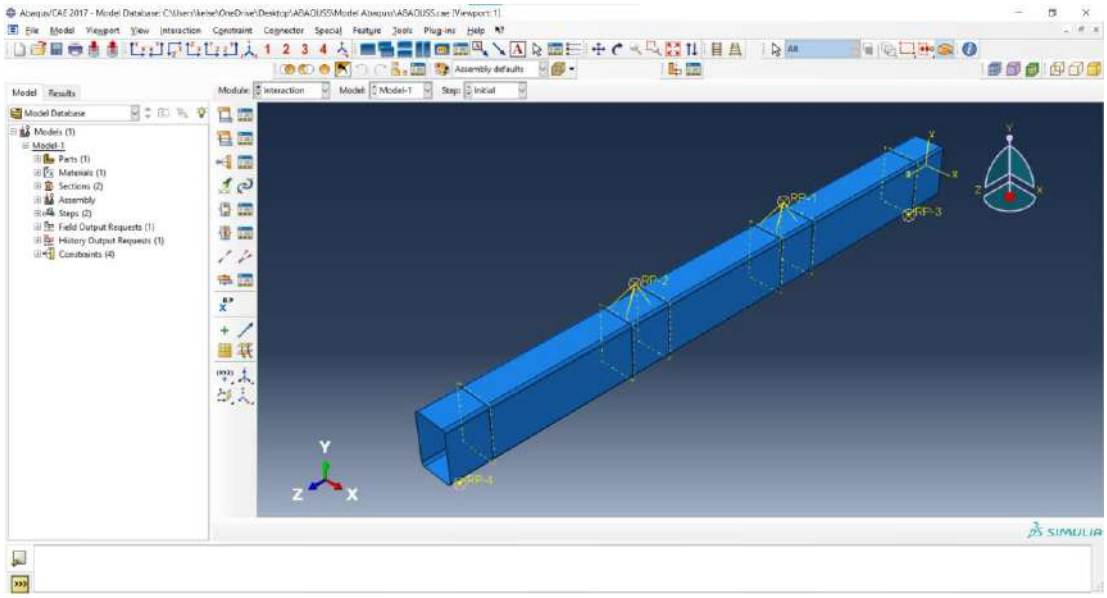




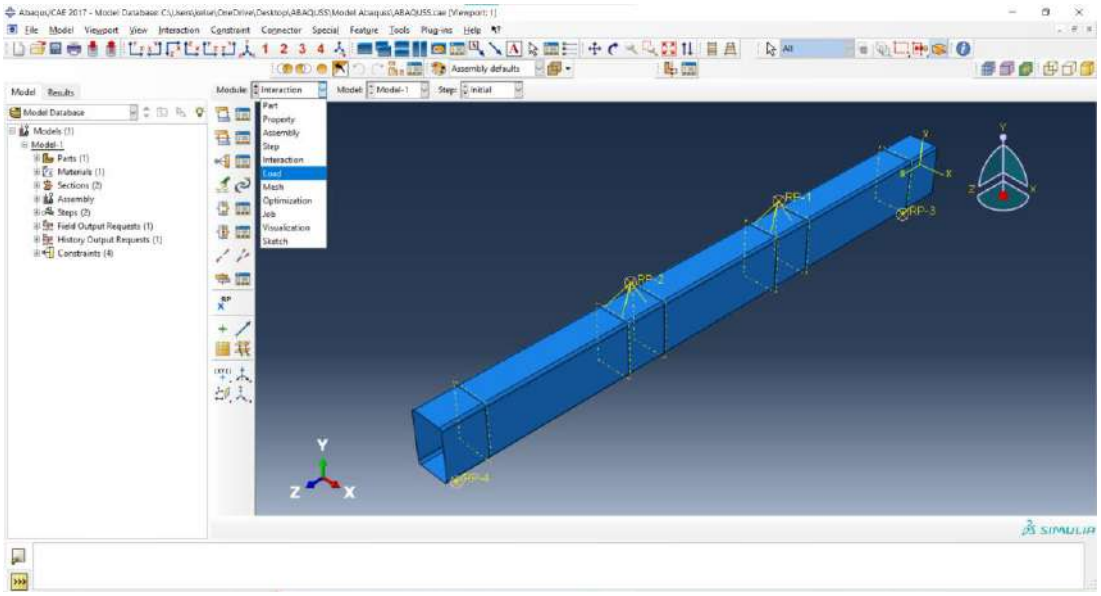




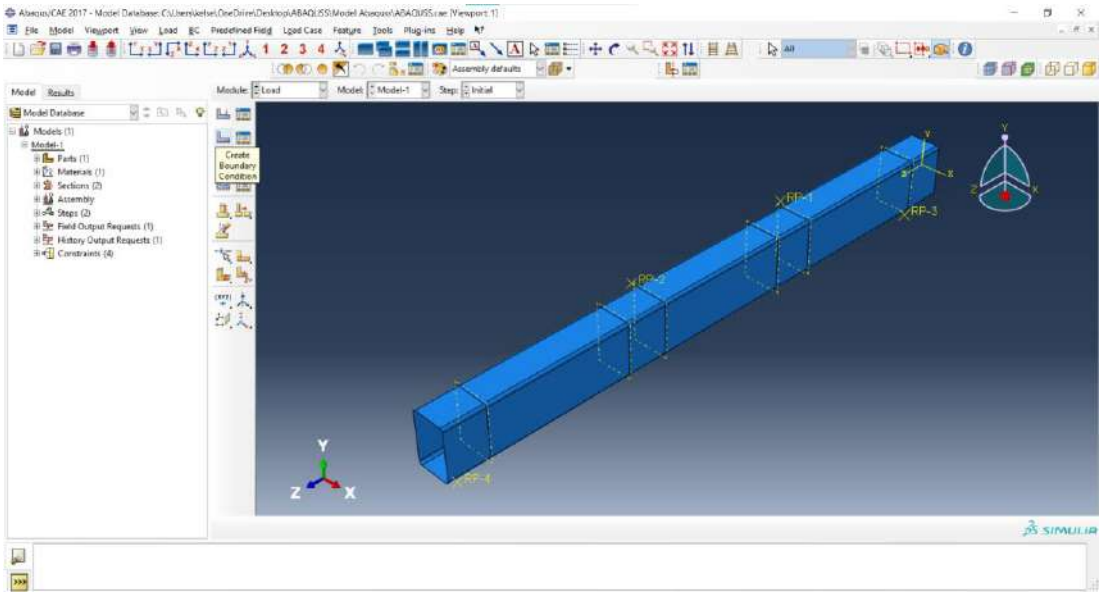
Lakukan hal yang sama untuk ketiga RP lainnya.

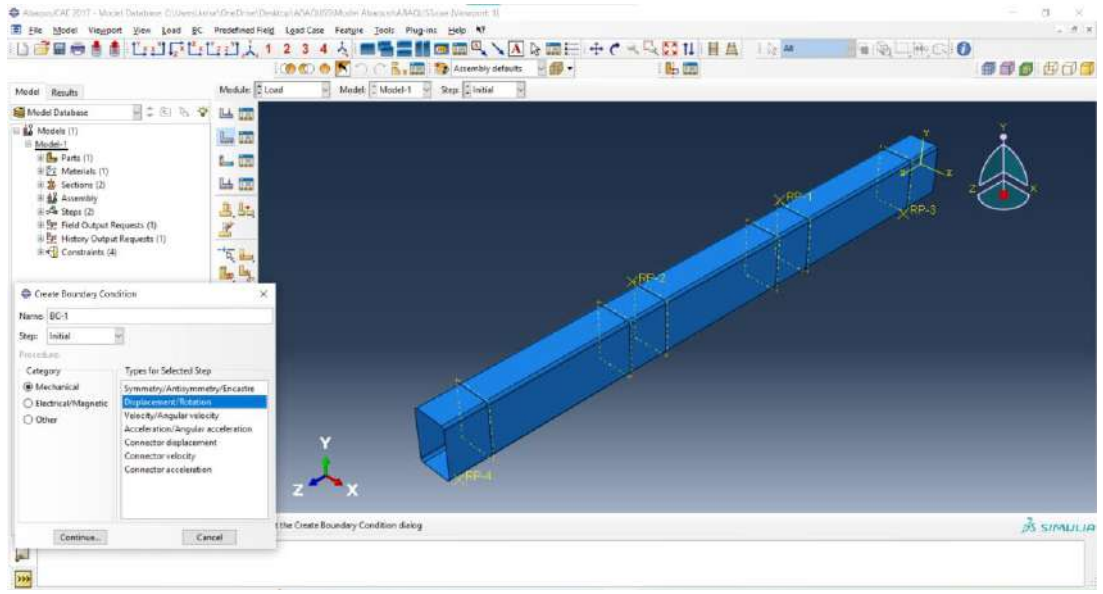


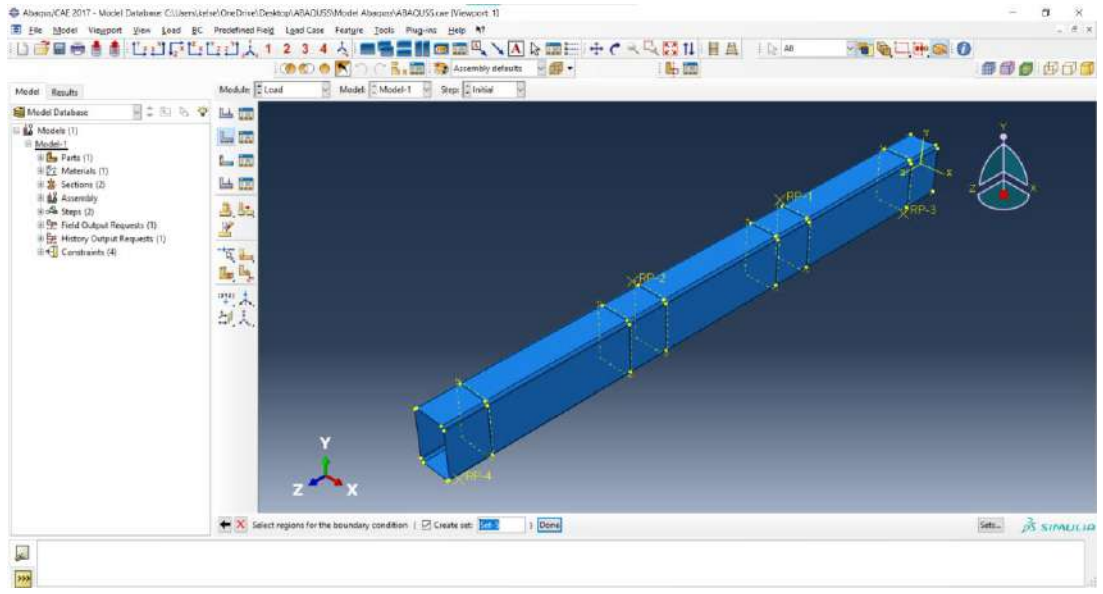
7. Membuat *Load*, pada *Module:* Pilih *Load*.

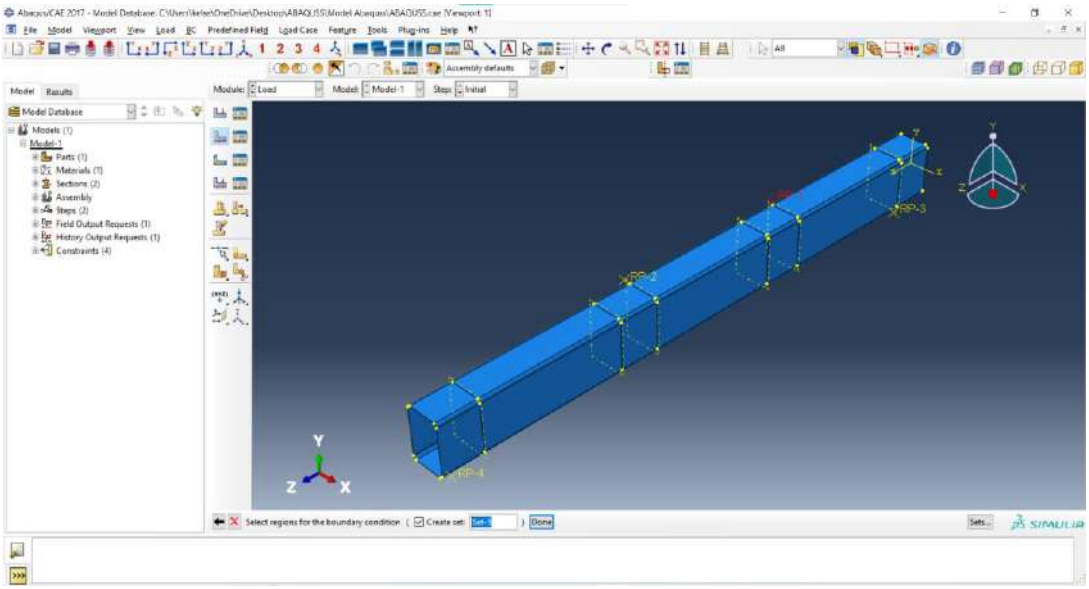


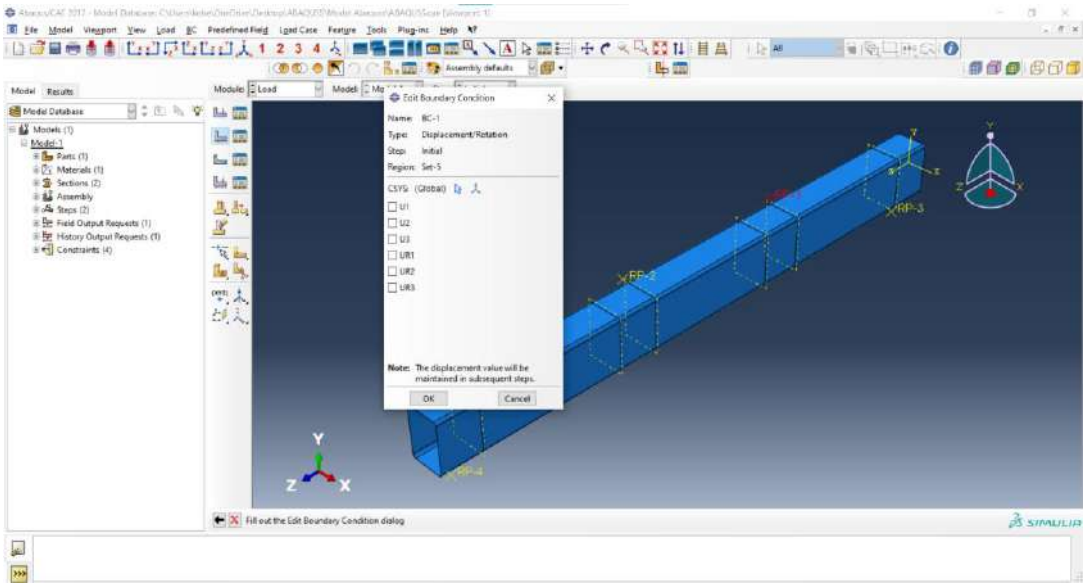
Klik *Create Boundary Condition* – Pada *Create Boundary Condition, Step: Initial, Types for Selected Step – Displacement/Rotation* – Klik *Continue*… – Klik RP-1 – Klik *Done* – Pada *Edit Boundary Condition*, Centang U1, UR2 dan UR3 – Klik Ok.

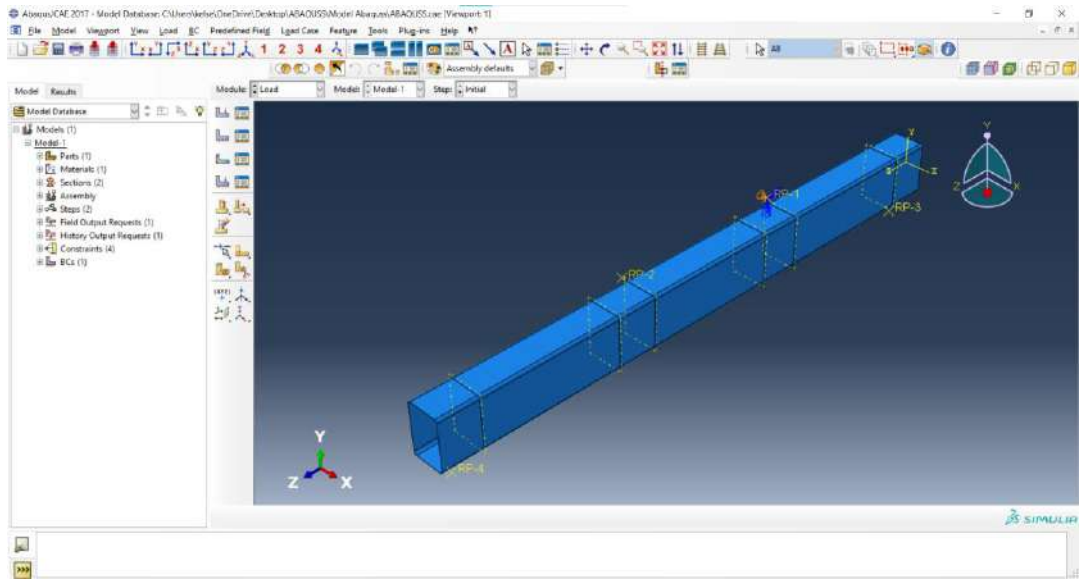

RP-1
RP-2
RP-3
RP-4
Y
Z
X


Create Boundary Condition
Name: BC-1
Step: Initial
Category
Mechanical
Electrical/Magnetic
Other
Types for Selected Step
Symmetry/Antisymmetry/Encastre
Displacement/Rotation
Velocity/Angular velocity
Acceleration/Angular acceleration
Connector displacement
Connector velocity
Connector acceleration
Continue...
Cancel
RP-1
RP-2
RP-3
RP-4


RP-1
RP-2
RP-3
RP-4
Y
Z
X

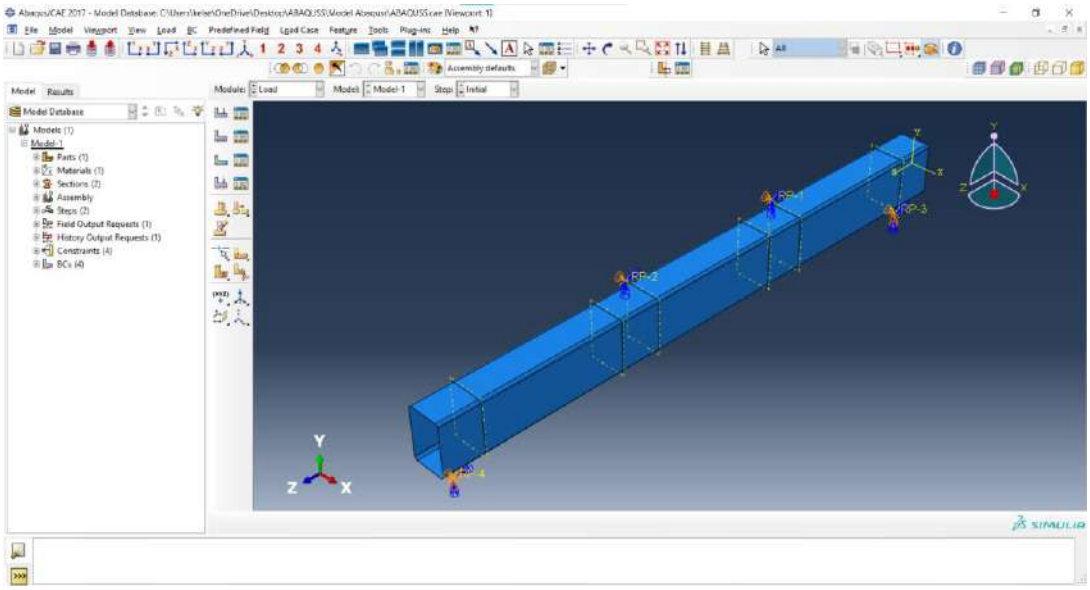




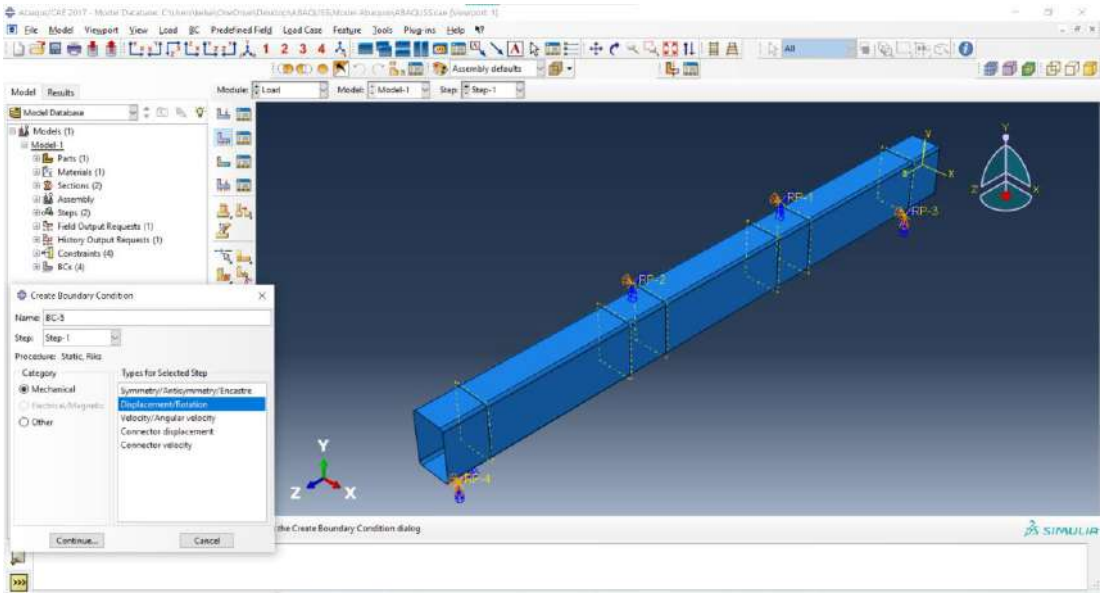


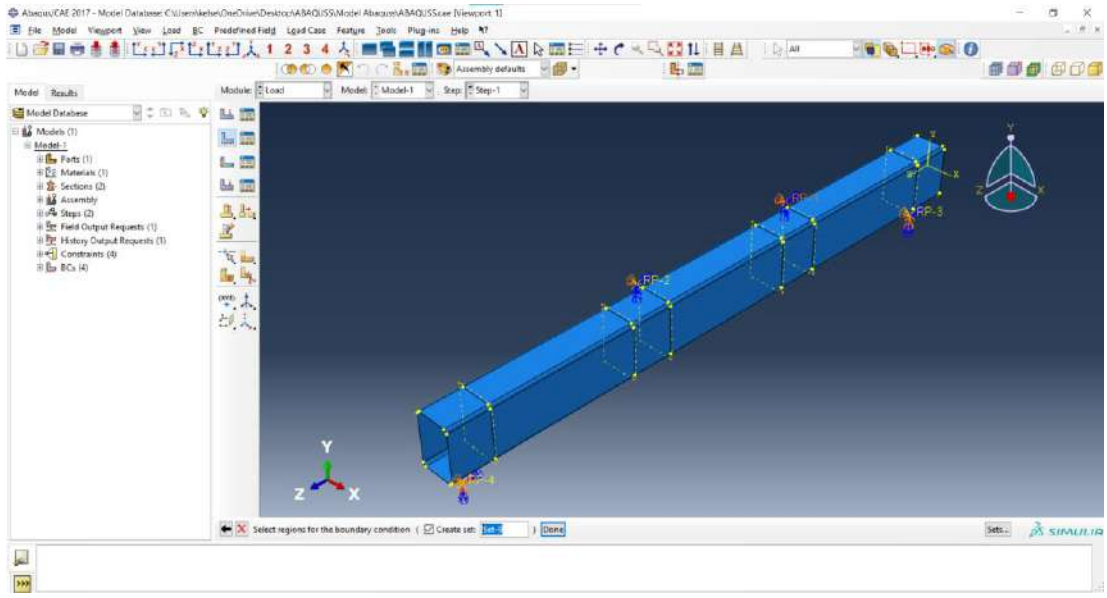
Lakukan hal yang sama untuk ketiga RP lainnya sesuai tabel di bawah ini.

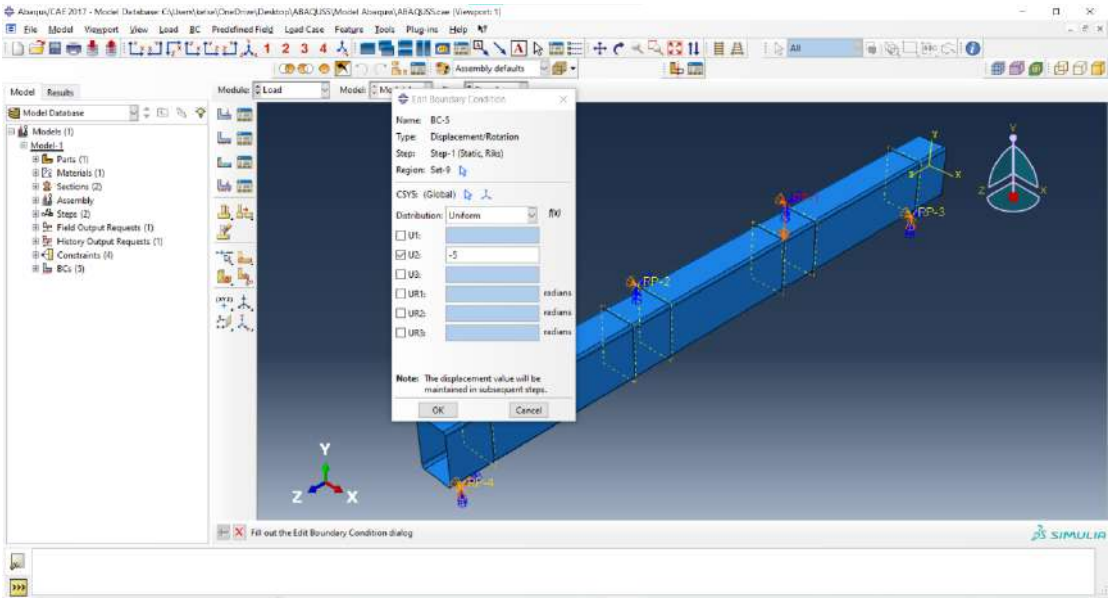
	U1	U2	U3	UR1	UR2	UR3
RP-1	✓				✓	✓
RP-2	✓				✓	✓
RP-3	✓	✓			✓	✓
RP-4	✓	✓	✓		✓	✓



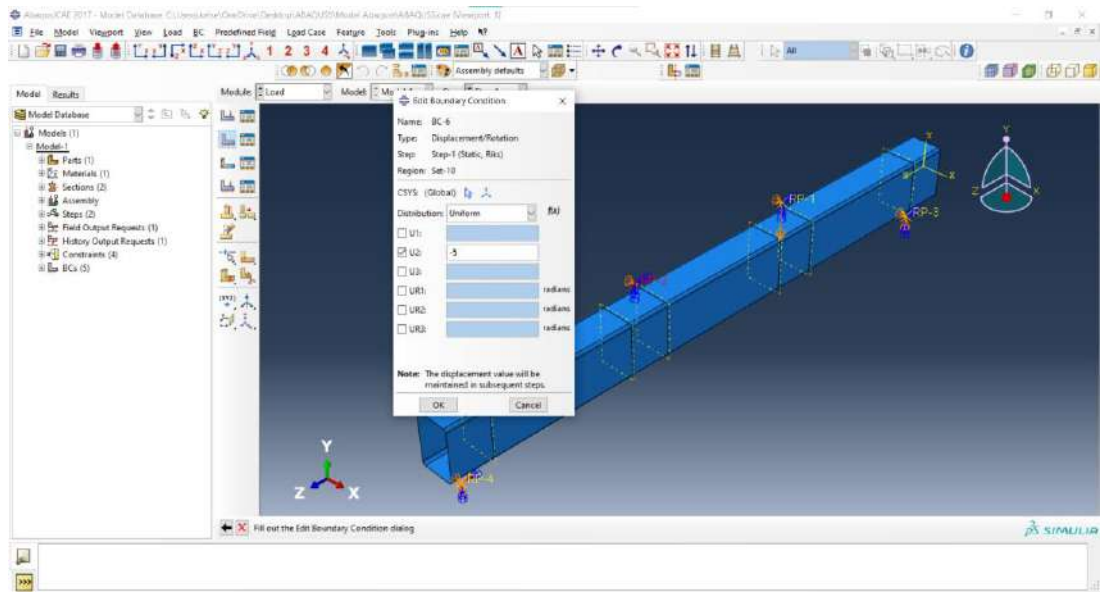
Klik *Create Boundary Condition* – Pada *Create Boundary Condition, Step: Step-1, Types for Selected Step – Displacement/Rotation* – Klik *Continue…* – Klik RP-1 – Klik *Done* – Pada *Edit Boundary Condition*, Centang U2, Input -5 – Klik Ok.

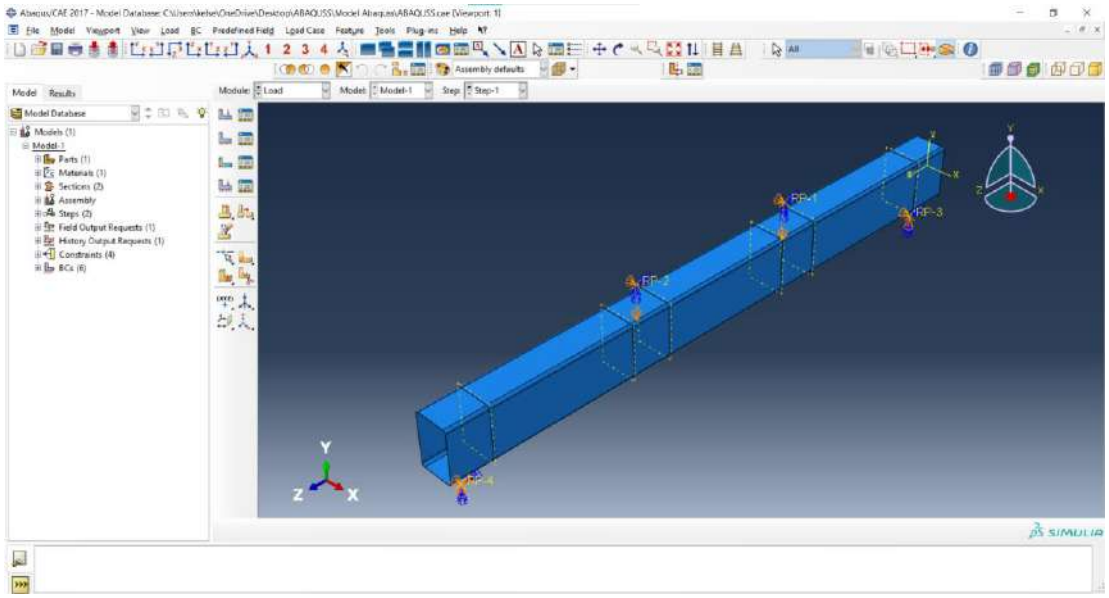




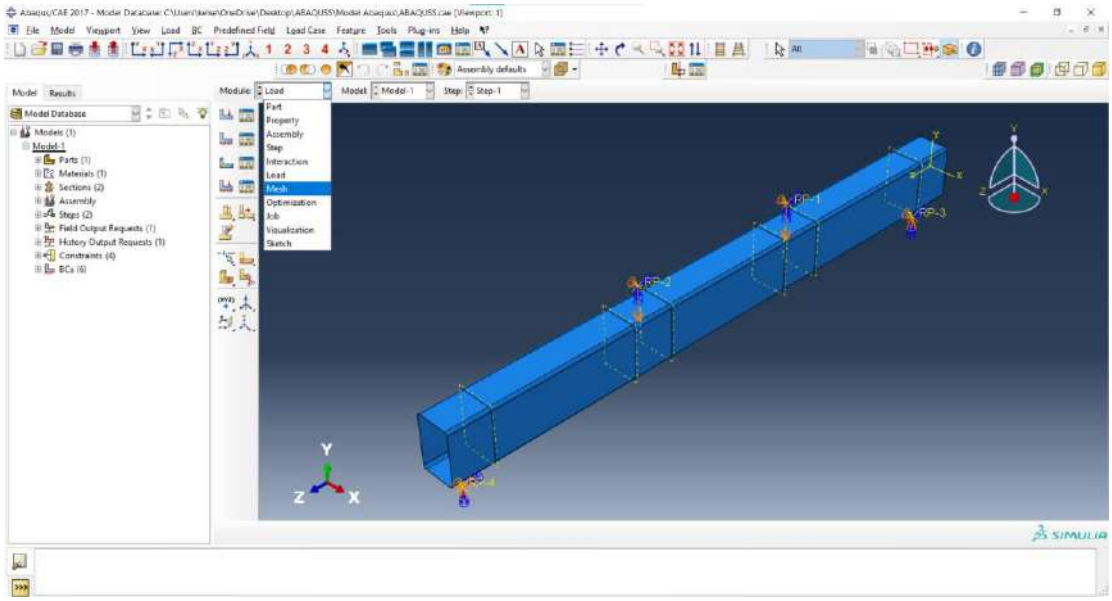


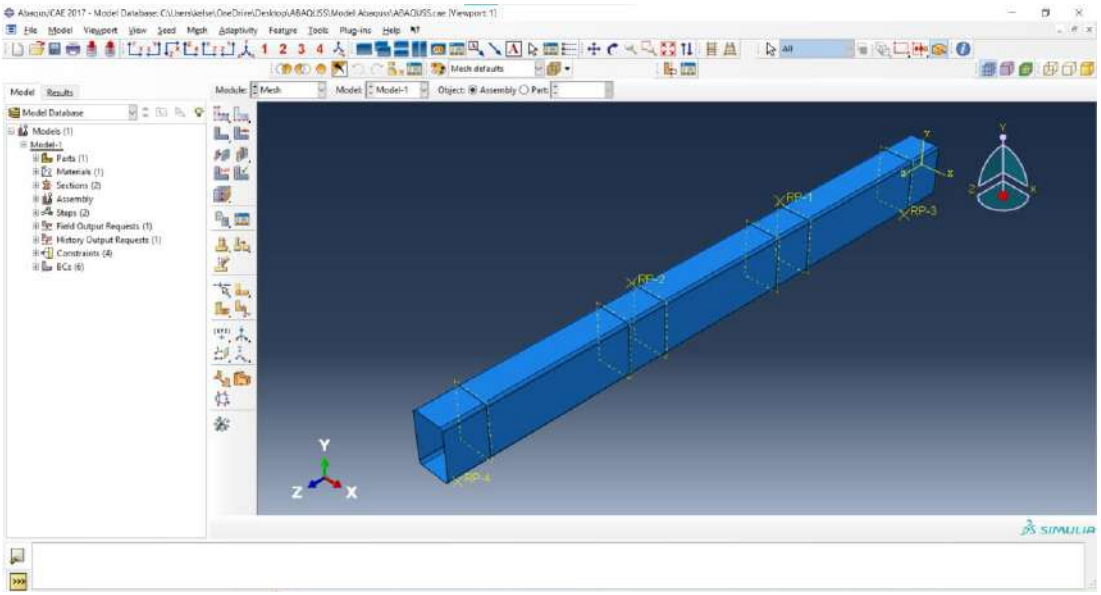
Lakukan hal yang sama untuk RP-2.



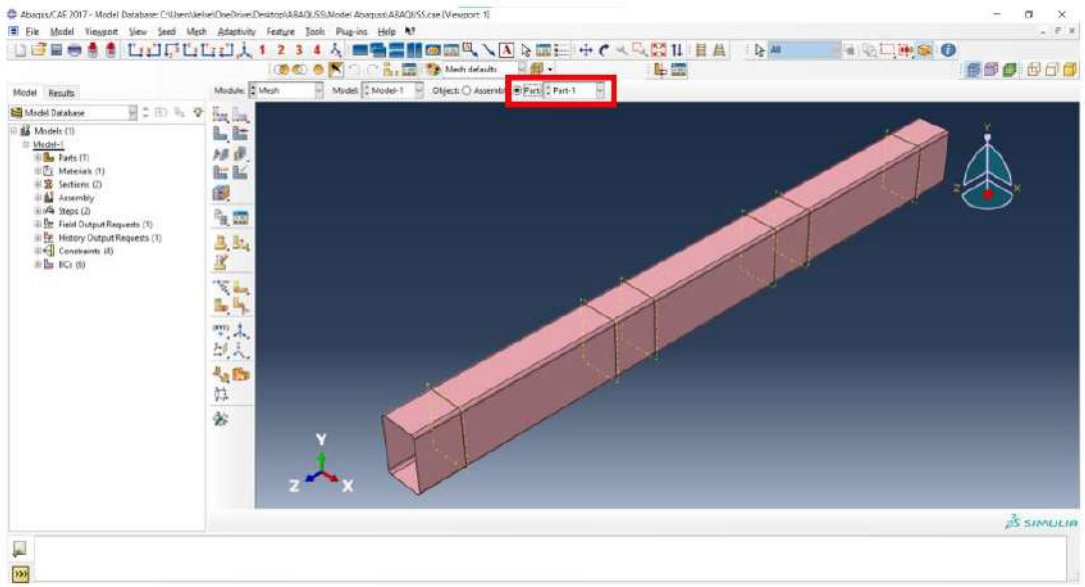


8. Membuat *Mesh*, pada *Module:* Pilih *Mesh*.

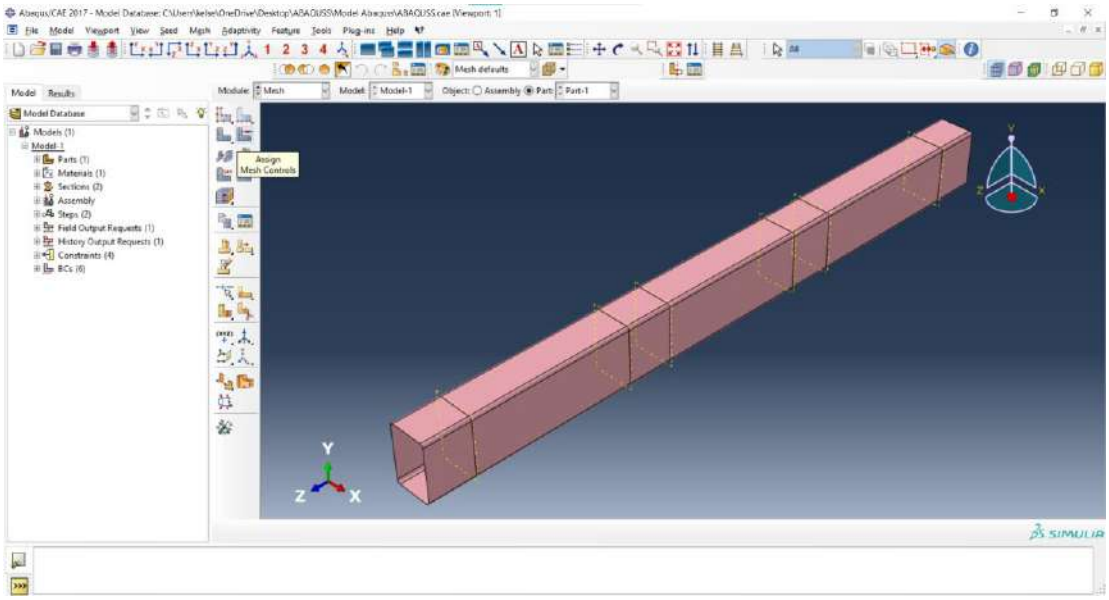


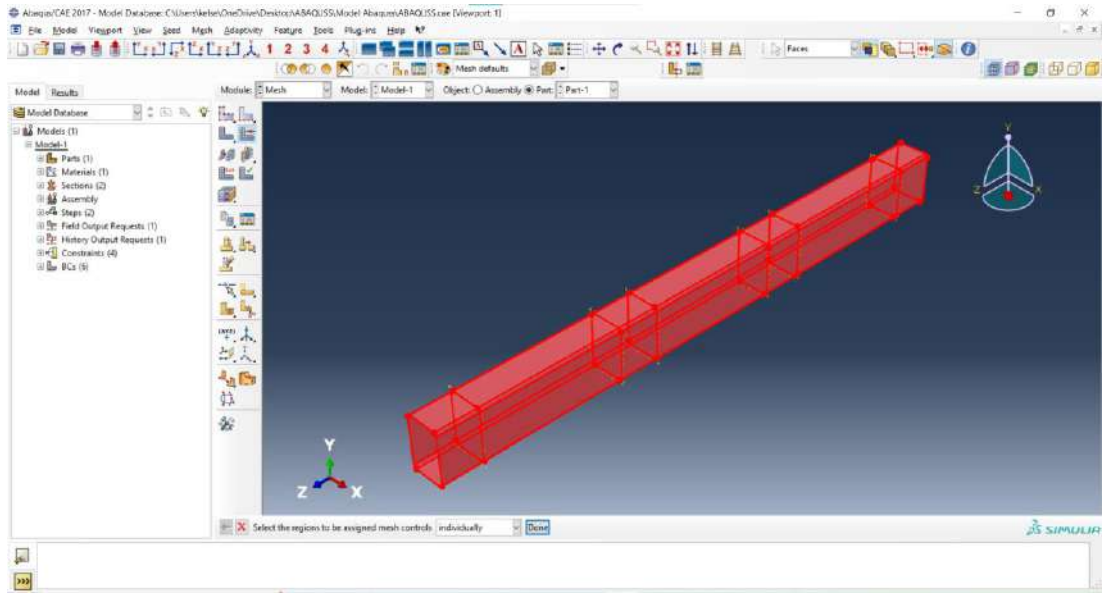


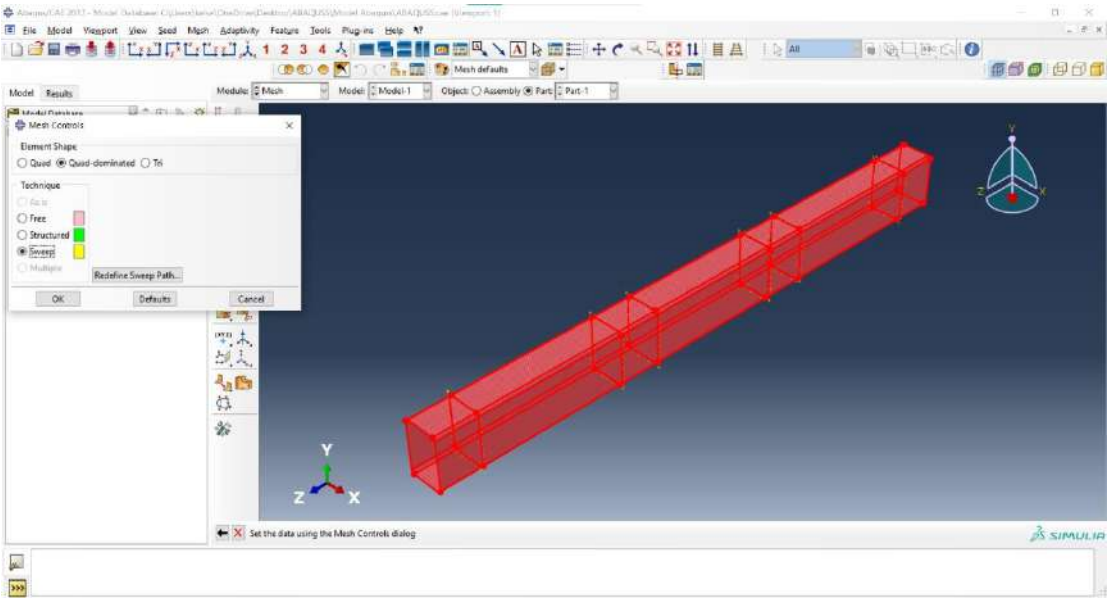
Pada *Object*, Klik *Part*

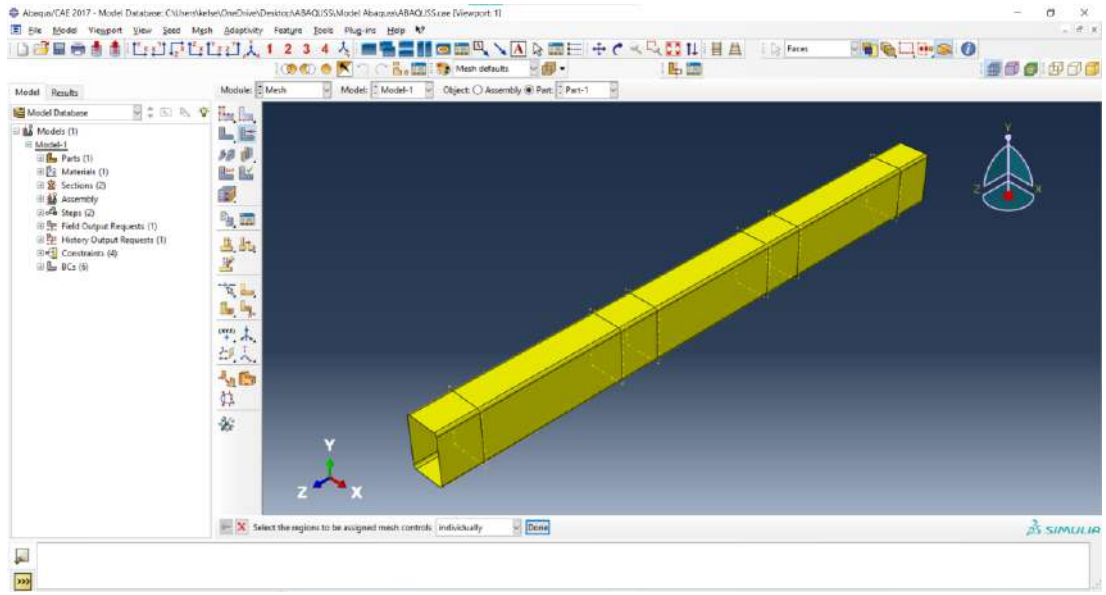


Klik *Assign Mesh Controls* – *Select* semua – Klik *Done* – Pada *Mesh Controls*, *Element Shape Quad-dominated*, *Technique Sweep* – Klik Ok.

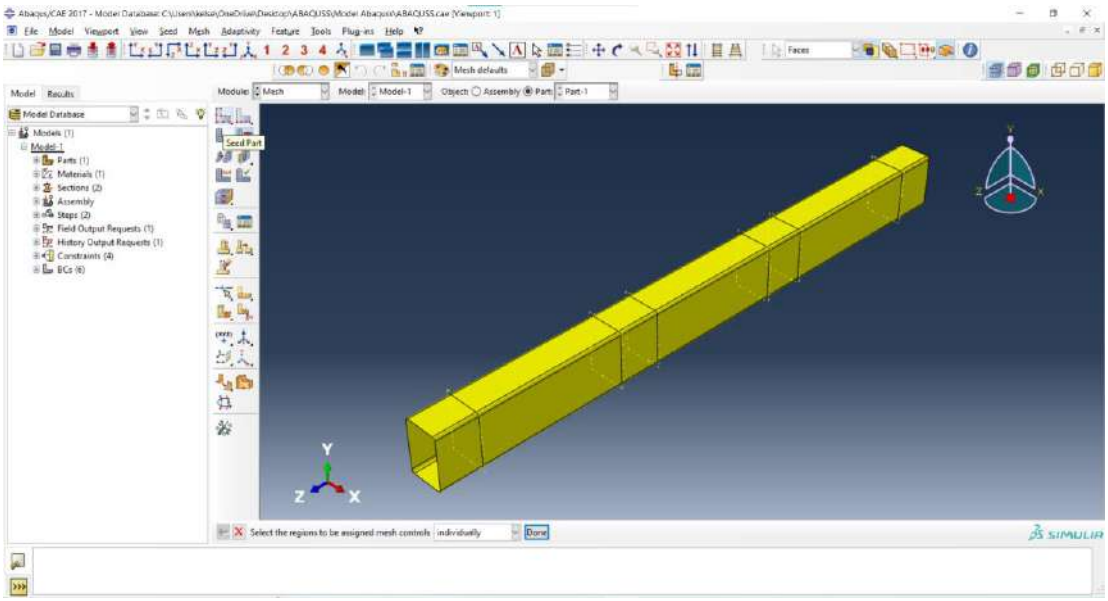


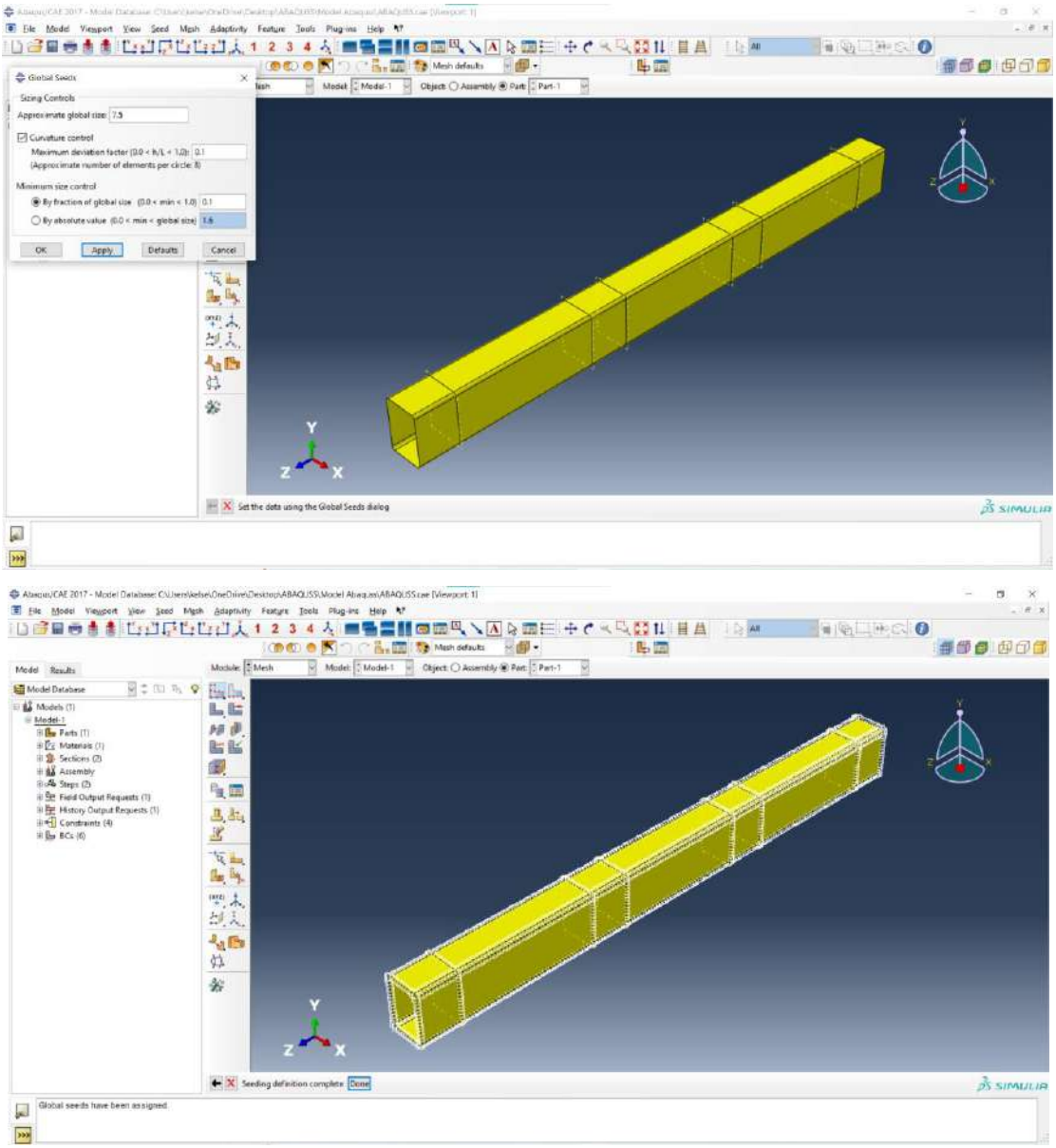



Mesh Controls
Element Shape
Quad
Quad-dominated
Tri
Technique
Free
Structured
Sweep
Redefine Sweep Path...
OK
Defaults
Cancel

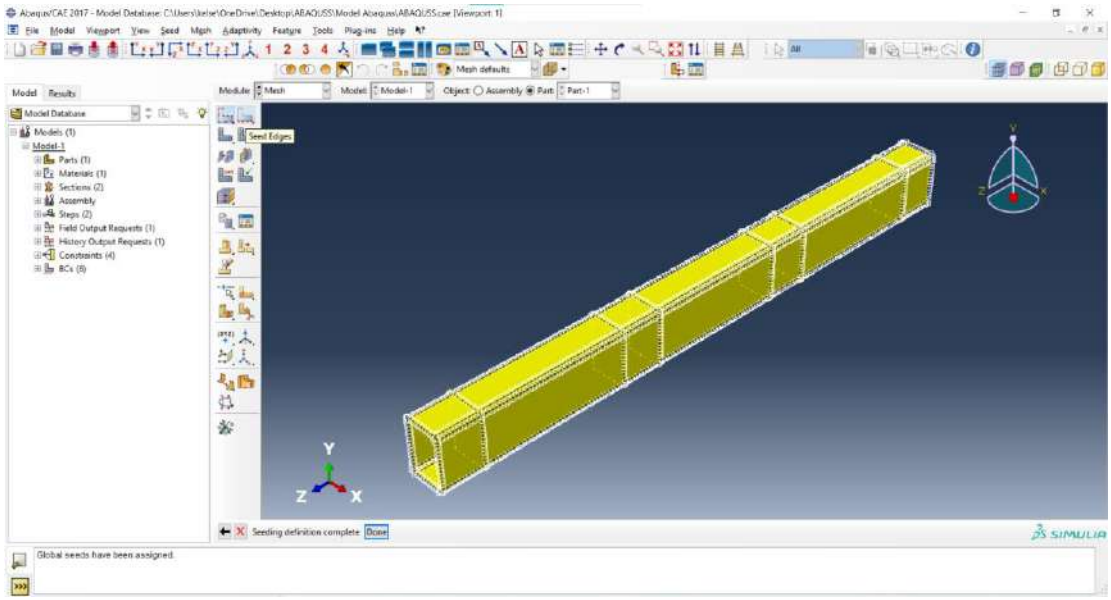


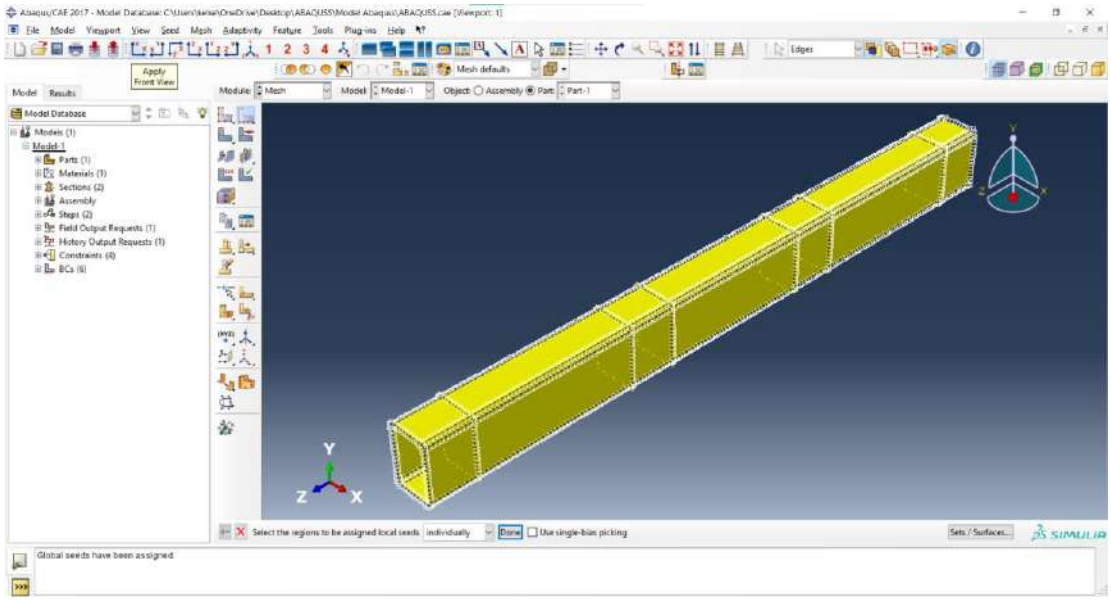
Klik *Seed Part* – Pada *Global Seeds*, *Sizing Controls*, *Approximate Global Size:* Input 7.5 – Klik Ok.

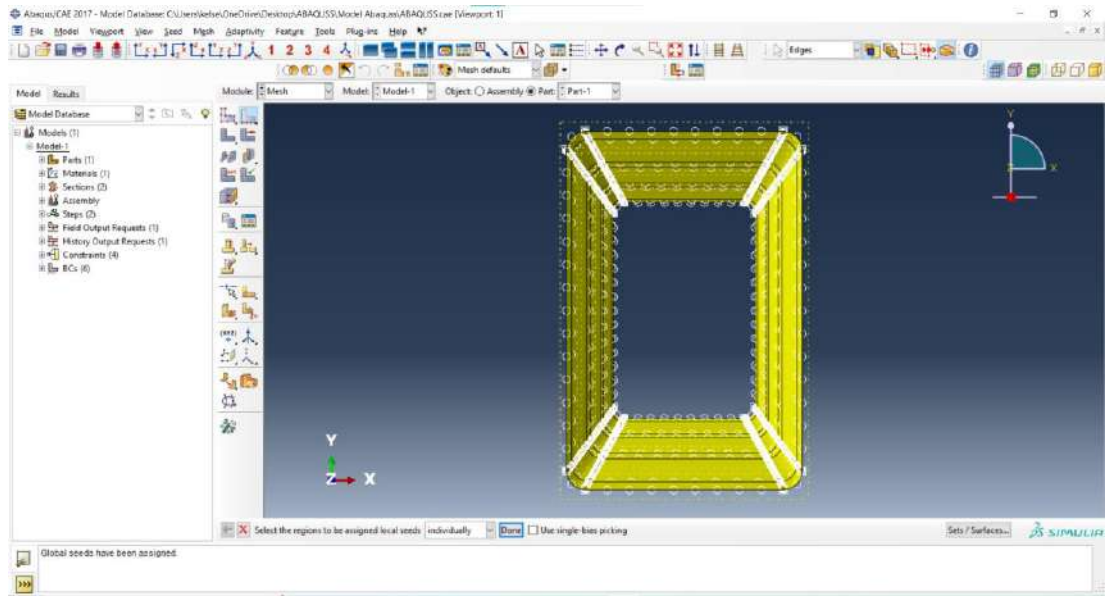


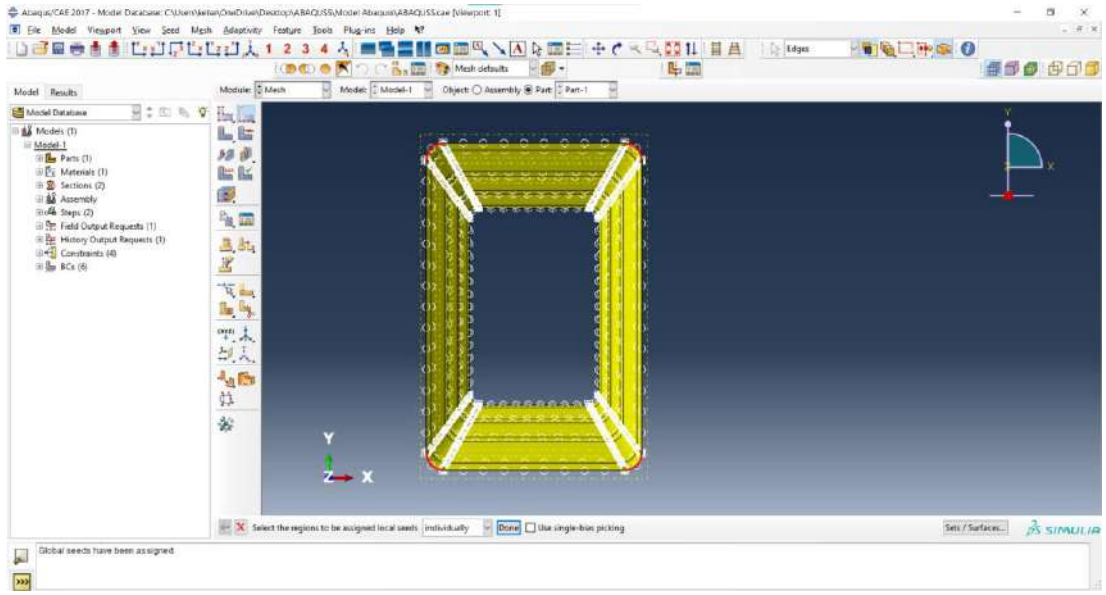


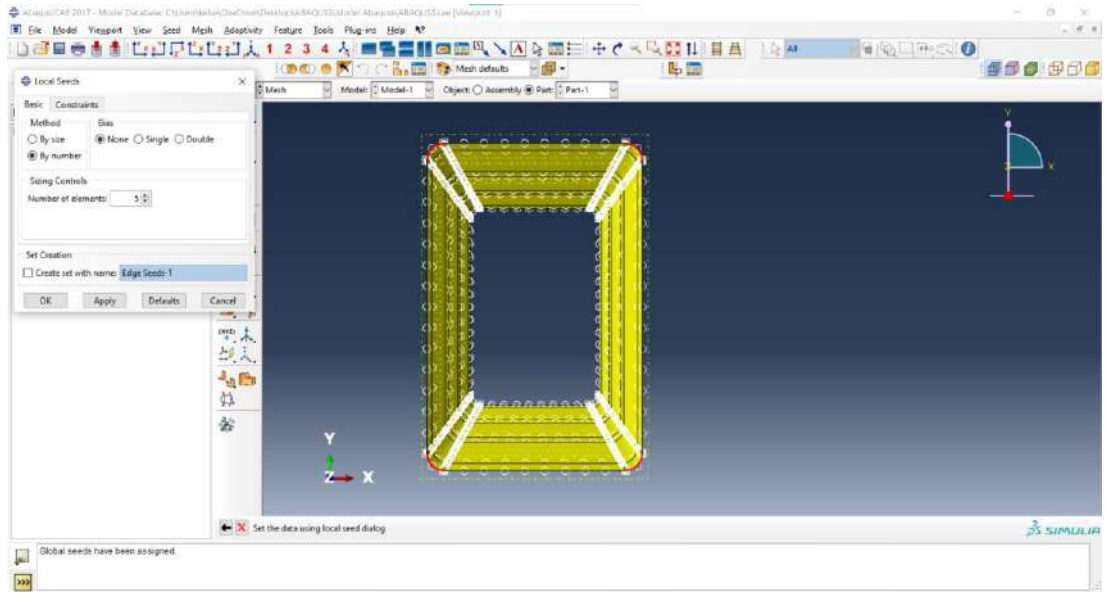
Klik *Seed Edges* – Klik *Apply Front View* – Klik Semua *Corner* dengan menahan tombol *Shift* saat mengklik *corner* – Klik *Done* – Pada *Local Seeds*, *Method By Number*, *Sizing Controls*, *Number of elements* input 5 – Klik Ok

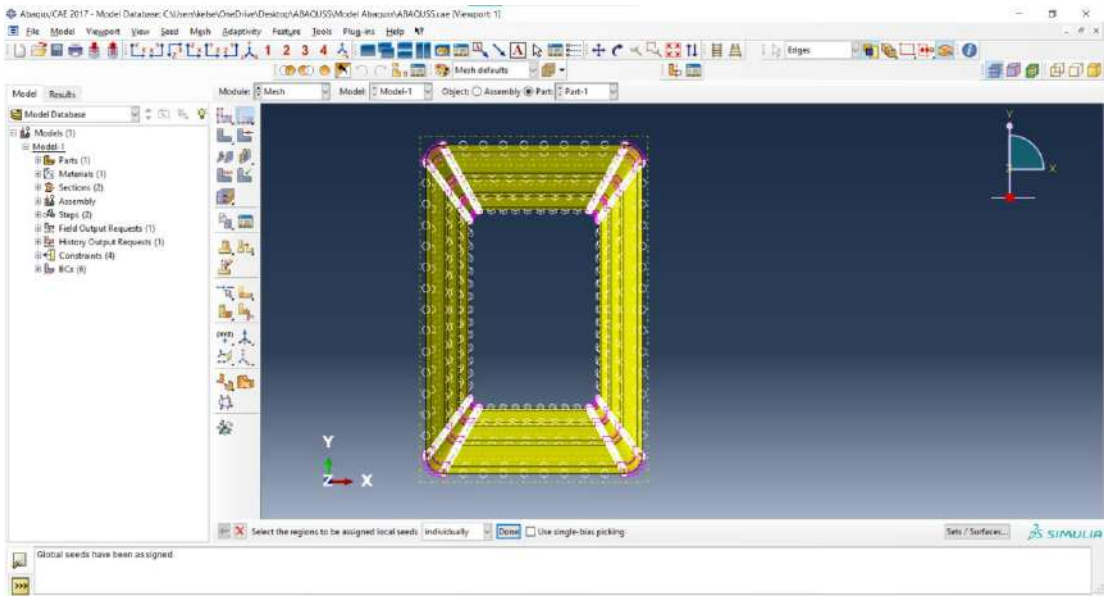




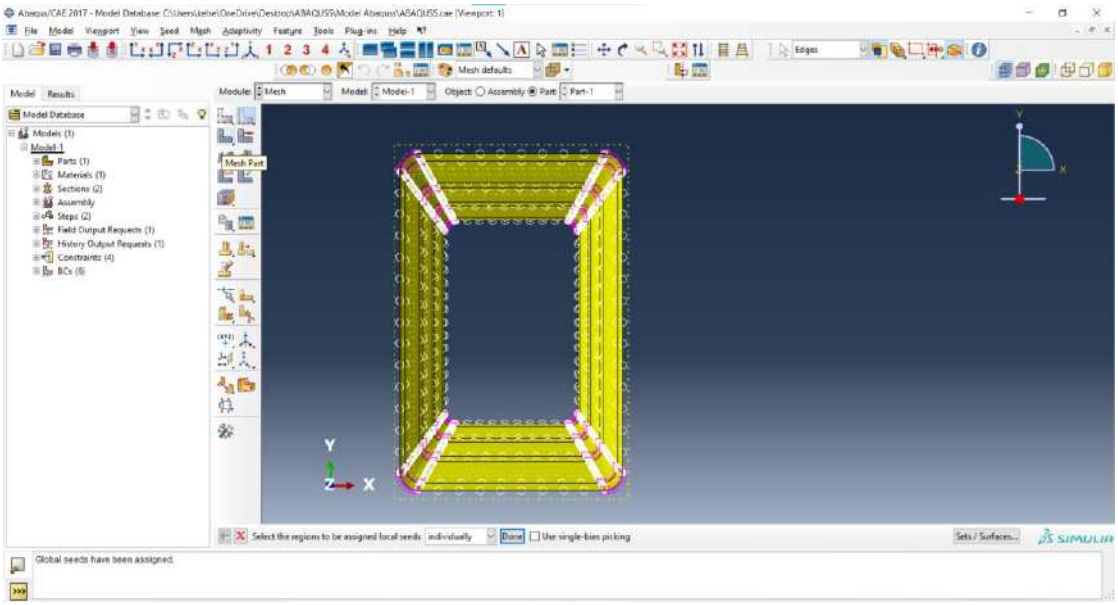


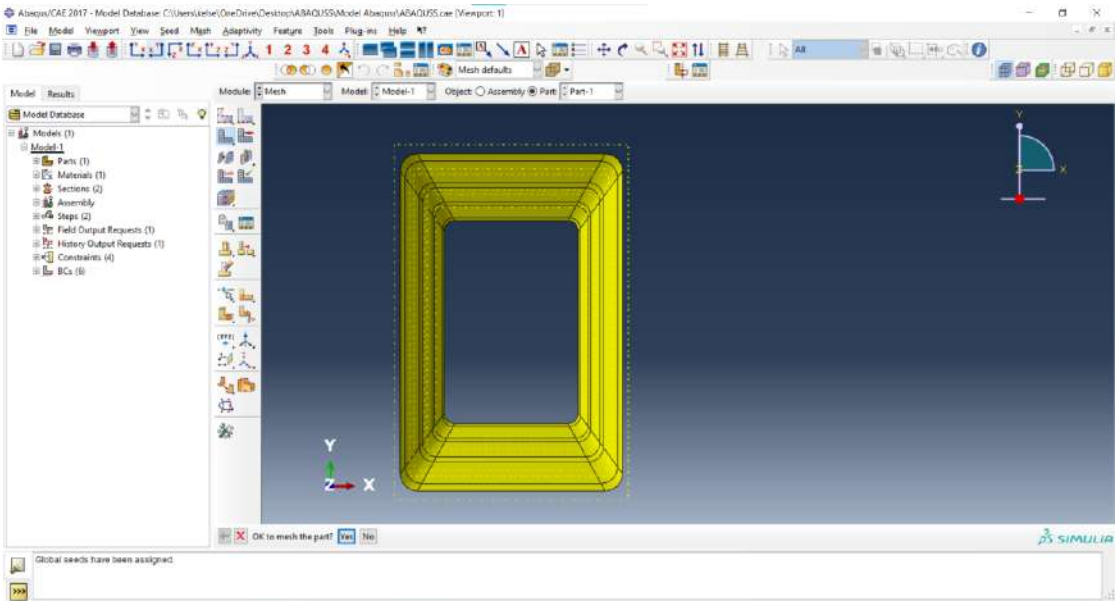


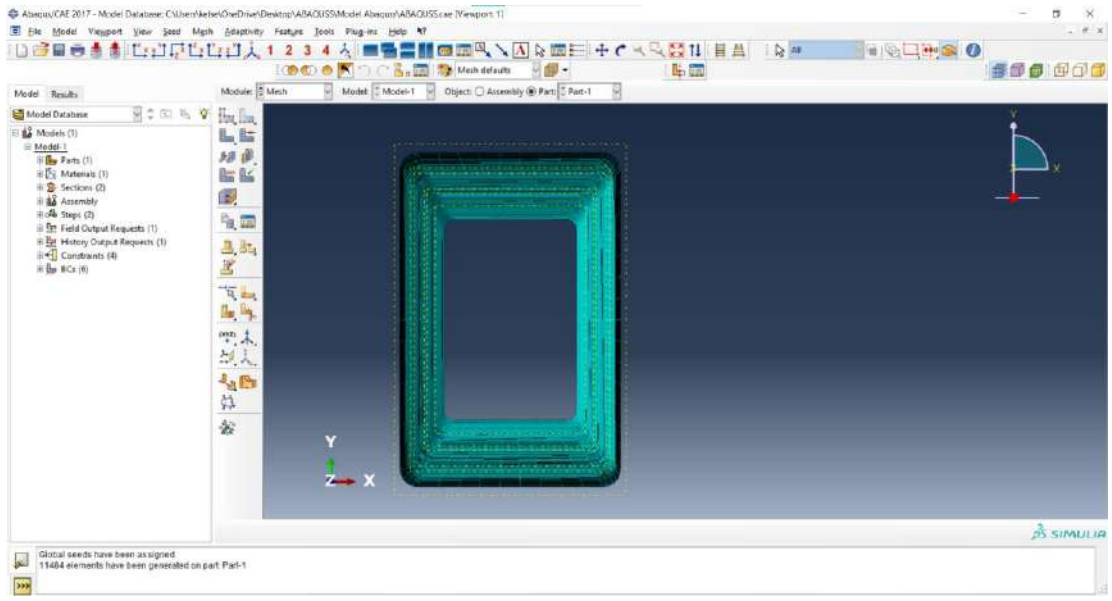


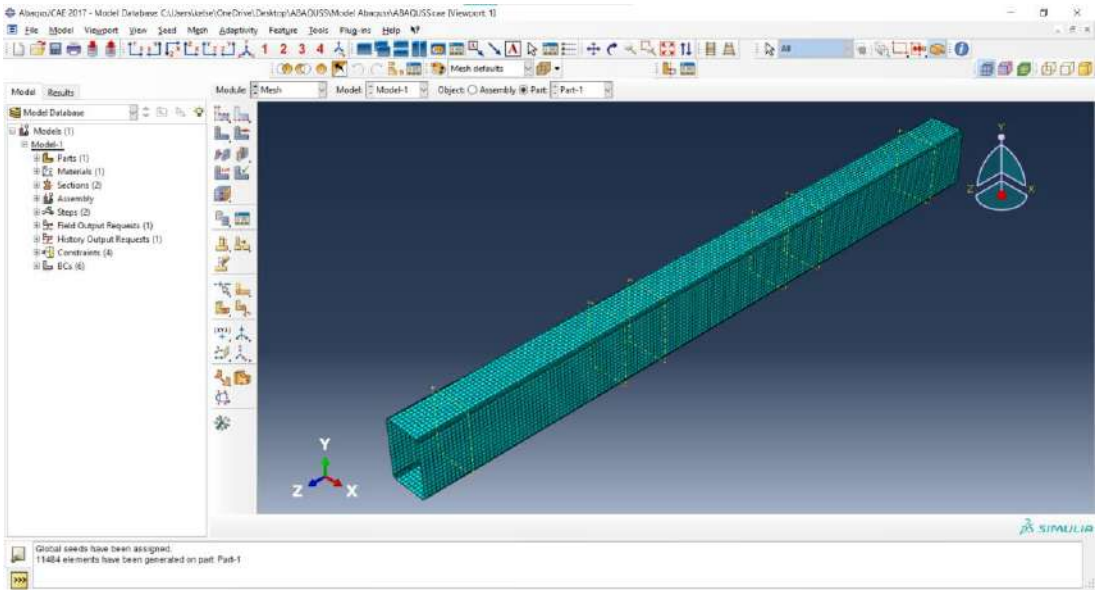


Klik *Mesh Part* – Klik *Yes.*

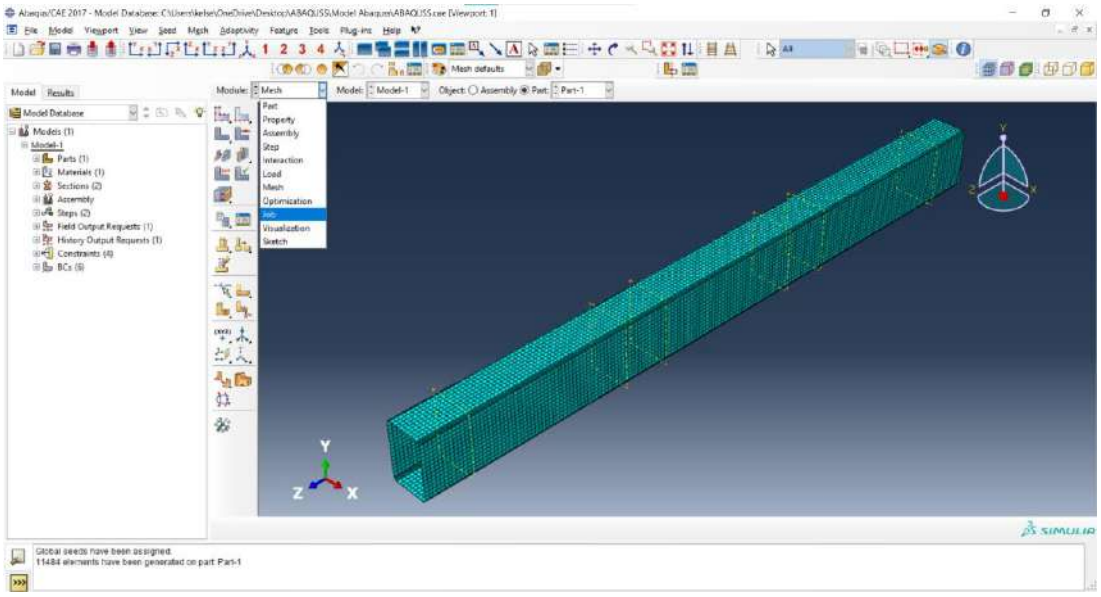

Global seeds have been assigned.

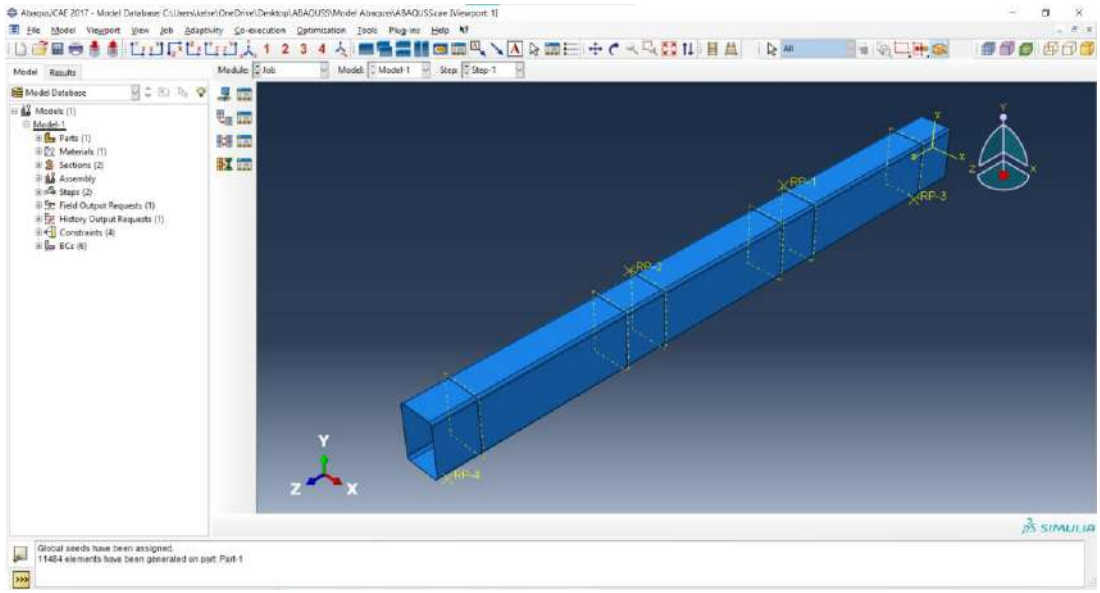

Global seeds have been assigned.


Global seeds have been assigned.
11484 elements have been generated on part: Part-1

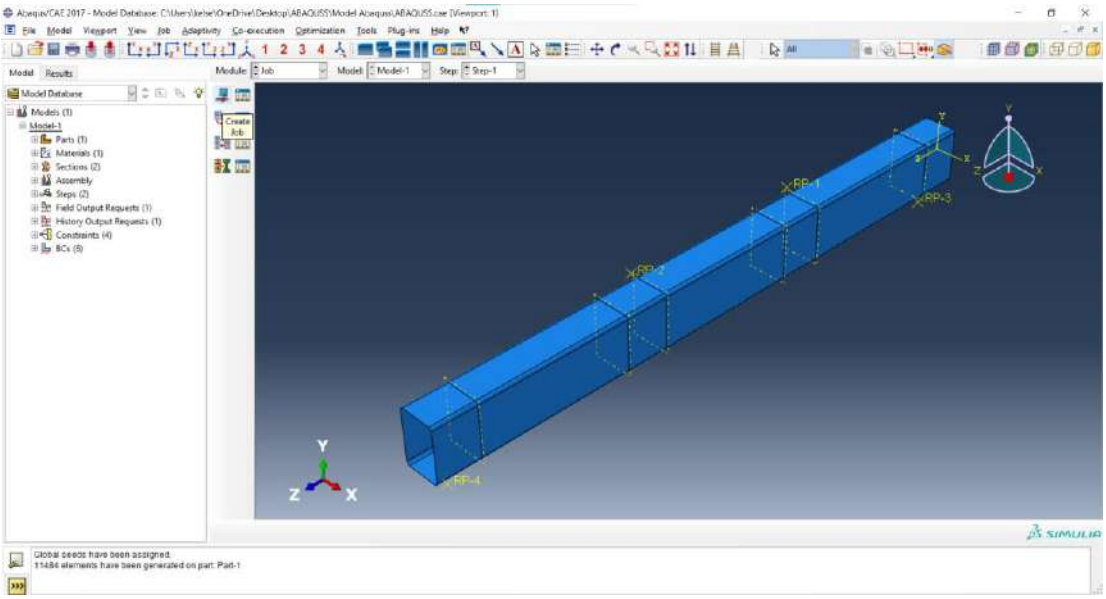


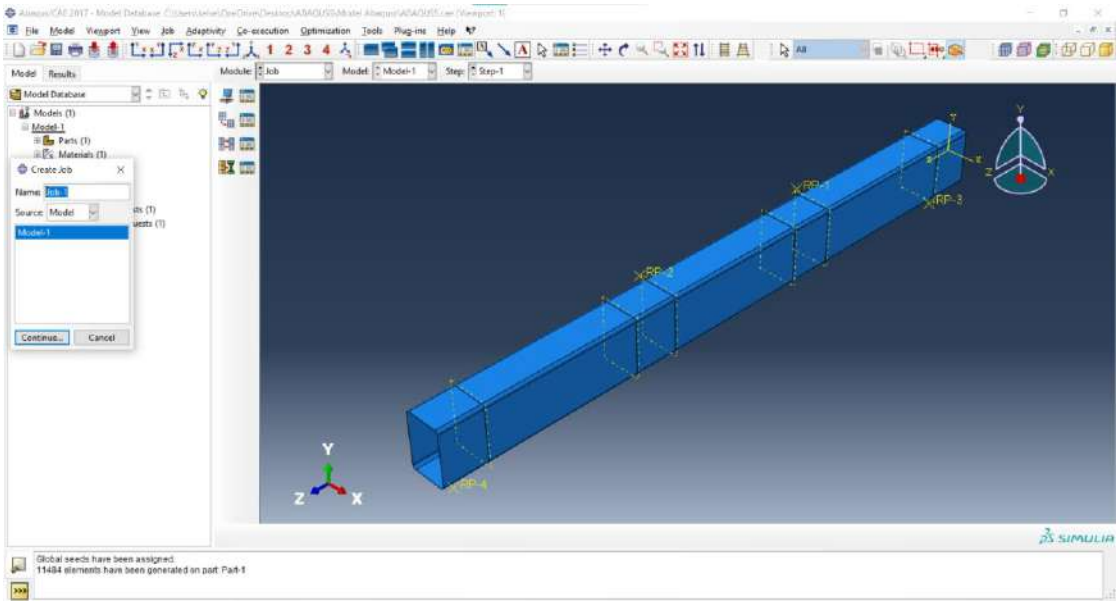
9. Membuat *Job*, pada *Module:* Pilih *Job*.

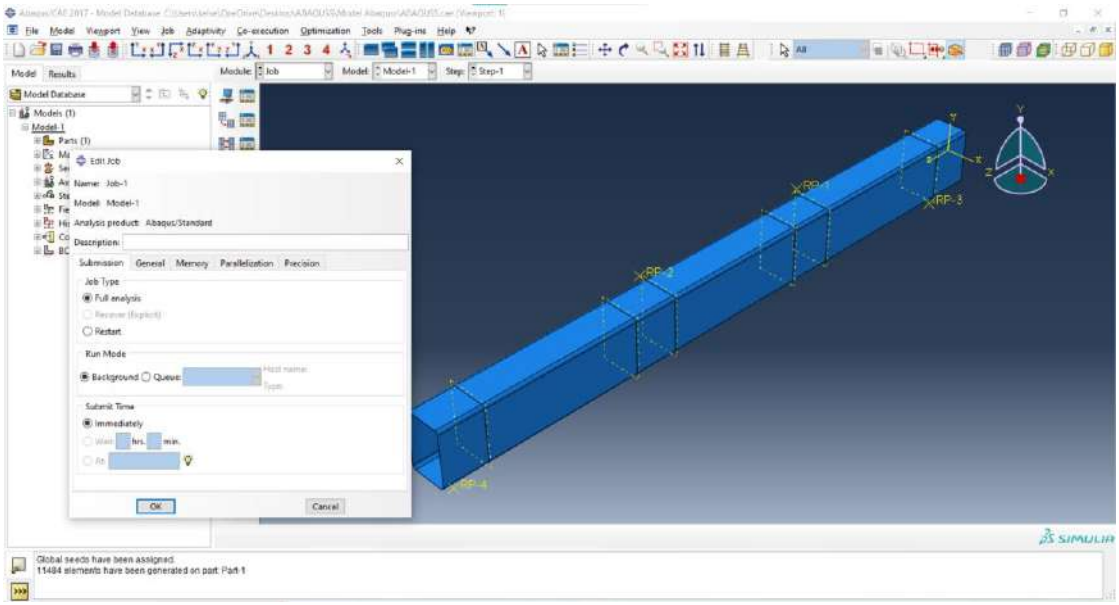


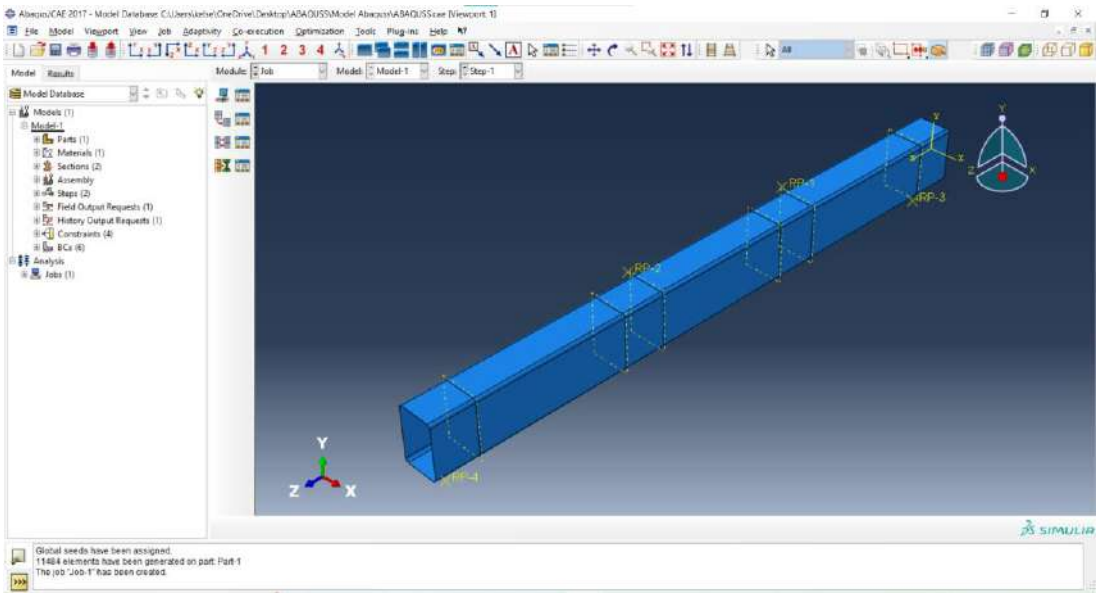


Klik *Create Job* – Pada *Create Job*, Klik *Continue*… – Pada *Edit Job*, Klik Ok.

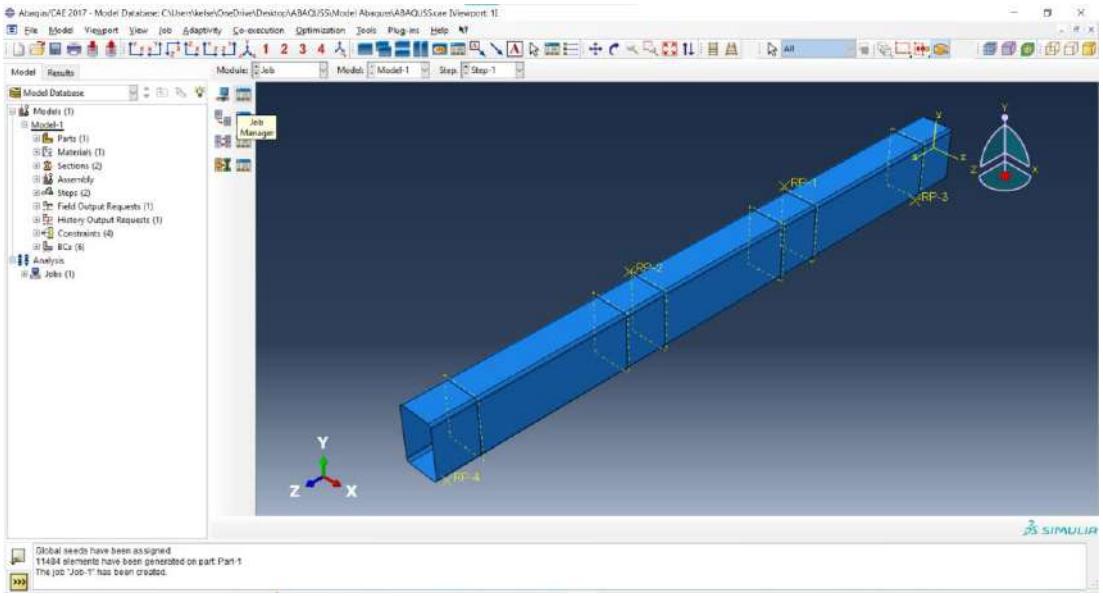


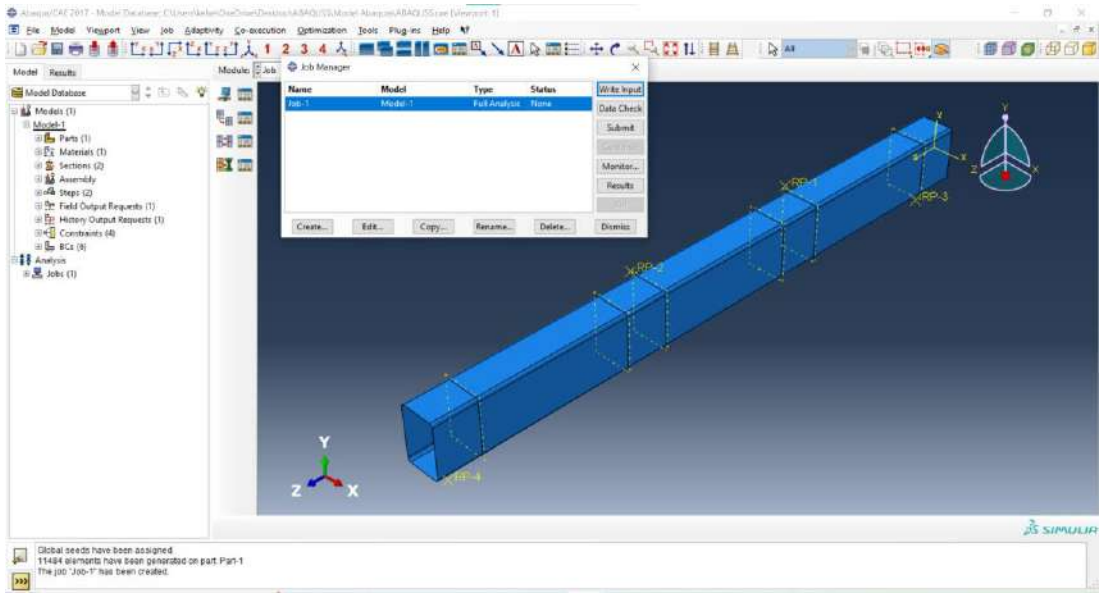

Create Job
Name: Job-1
Source: Model
Model-1
Continue...
Cancel
RP-1
RP-2
RP-3
RP-4
Y
Z
X
Global seeds have been assigned.
11484 elements have been generated on part: Part-1

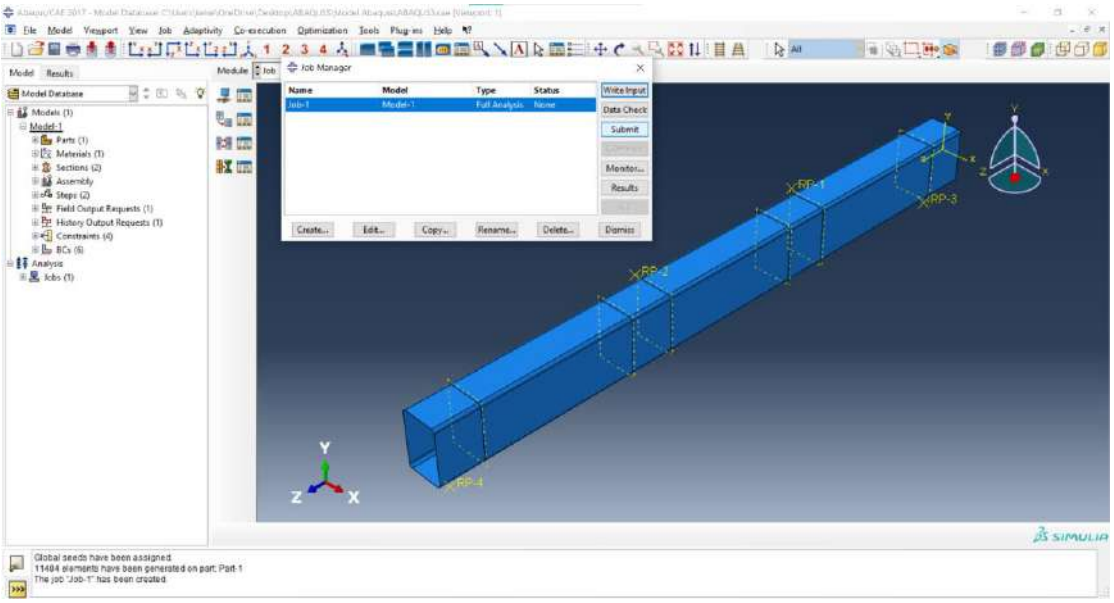

Edit Job
Name: Job-1
Model: Model-1
Analysis product: Abaqus/Standard
Description:
Submission
General
Memory
Parallelization
Precision
Job Type
Full analysis
Restart
Run Mode
Background
Queue
Immediately
OK
Cancel
RP-1
RP-2
RP-3
RP-4
Global seeds have been assigned.
11484 elements have been generated on part: Part-1

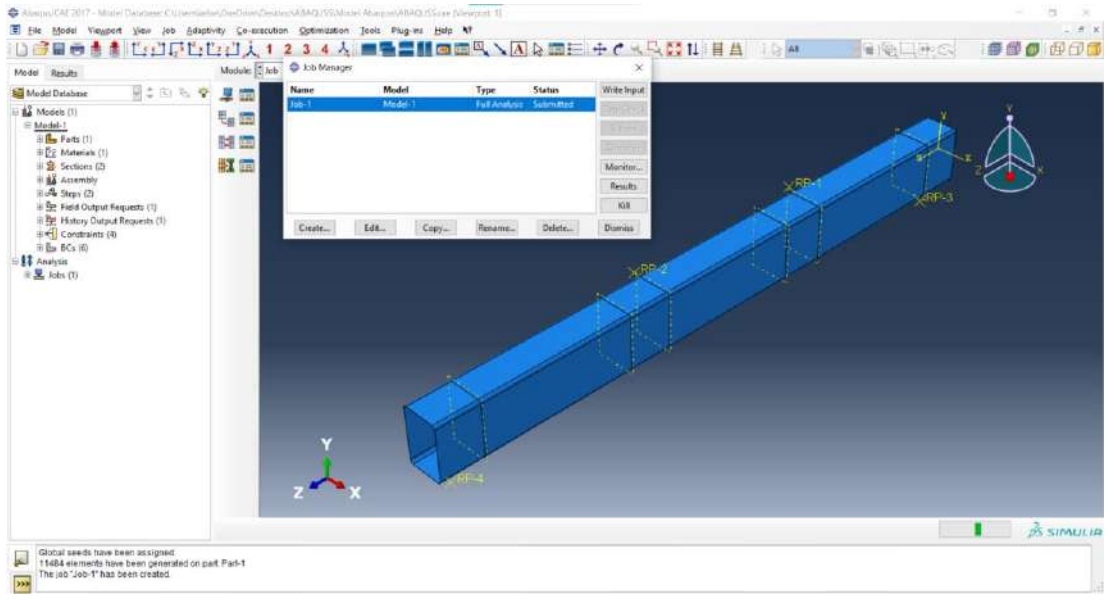

Model Database
Models (1)
Model-1
Parts (1)
Materials (1)
Sections (2)
Assembly
Steps (2)
Field Output Requests (1)
History Output Requests (1)
Constraints (4)
BCs (6)
Analysis
Jobs (1)
RP-1
RP-2
RP-3
RP-4
Global seeds have been assigned.
11484 elements have been generated on part: Part-1
The job "Job-1" has been created.

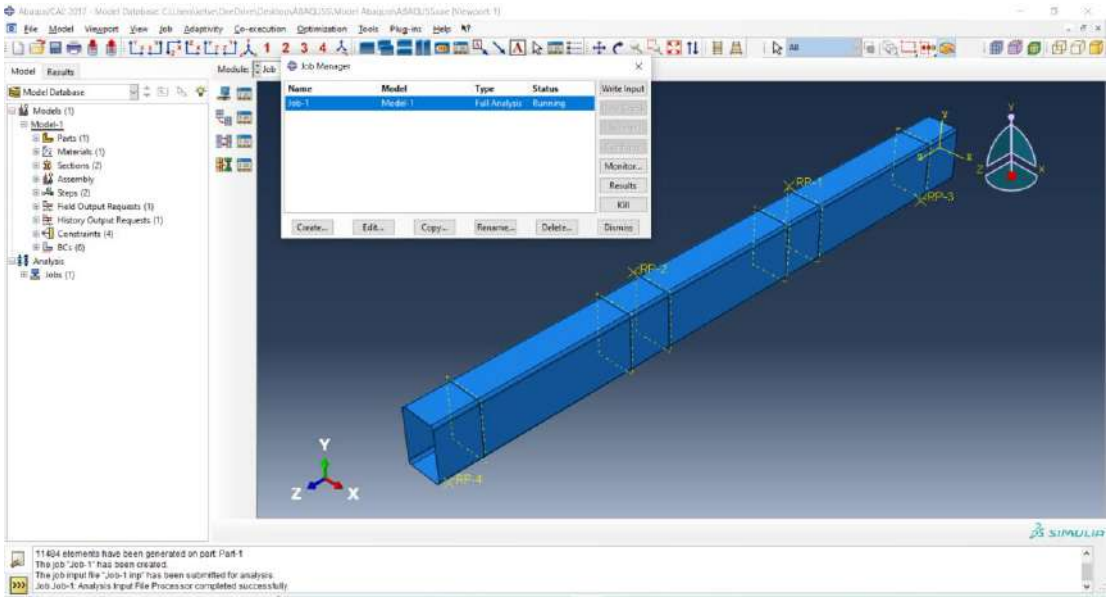
Klik *Job Manager* – Pada *Job Manager*, Klik *Submit* – Tunggu hingga selesai run.

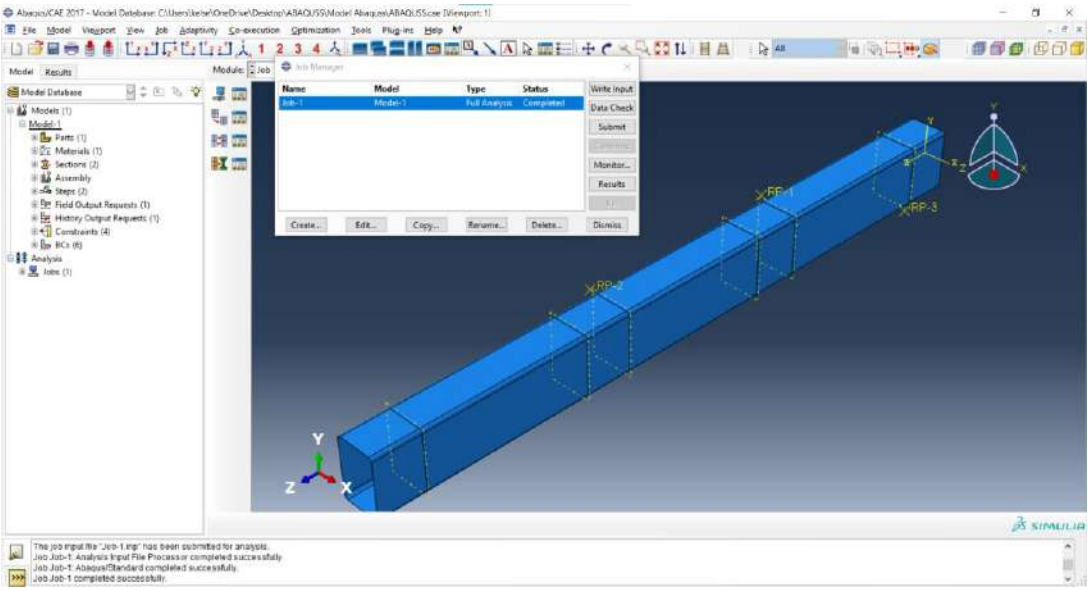




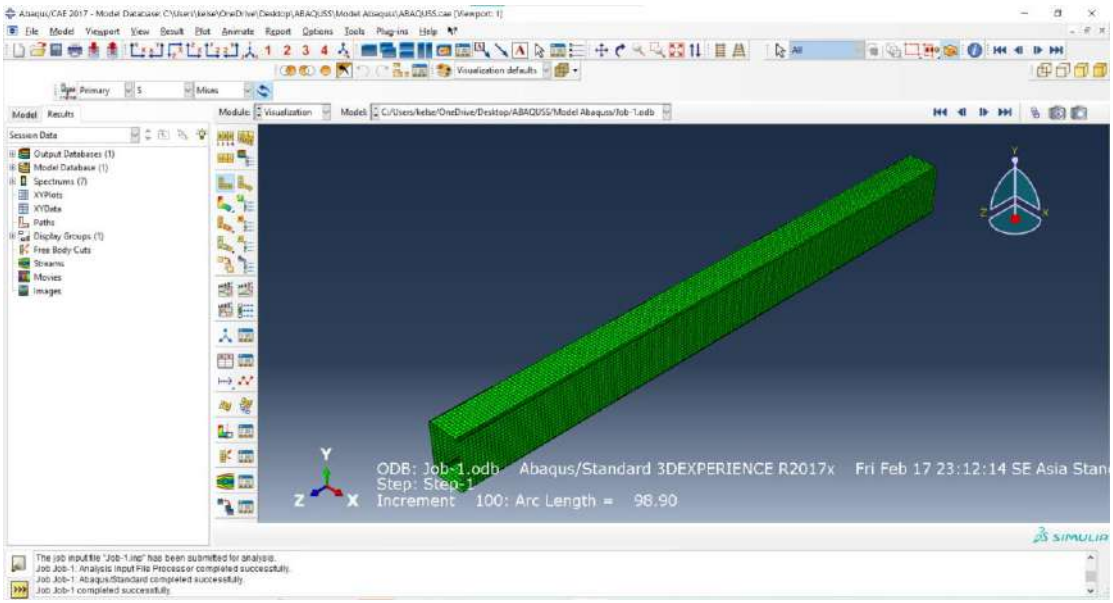

Job Manager
Name
Model
Type
Status
Job-1
Model-1
Full Analysis
None
Write Input
Data Check
Submit
Monitor...
Results
Create...
Edit...
Copy...
Rename...
Delete...
Dismiss
RP-1
RP-2
RP-3
RP-4
Global seeds have been assigned.
11484 elements have been generated on part: Part-1
The job "Job-1" has been created.

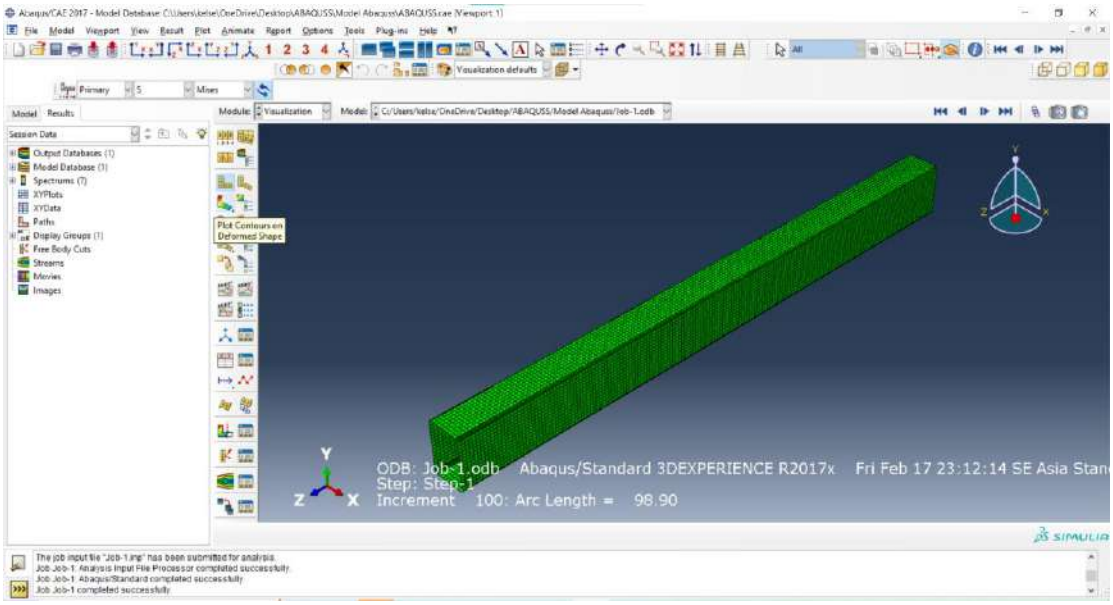

Job Manager
Name
Model
Type
Status
Job-1
Model-1
Full Analysis
Submitted
Write Input
Monitor...
Results
Kill
Create...
Edit...
Copy...
Rename...
Delete...
Dismiss
RP-1
RP-2
RP-3
RP-4
Global seeds have been assigned.
11484 elements have been generated on part: Part-1
The job "Job-1" has been created.

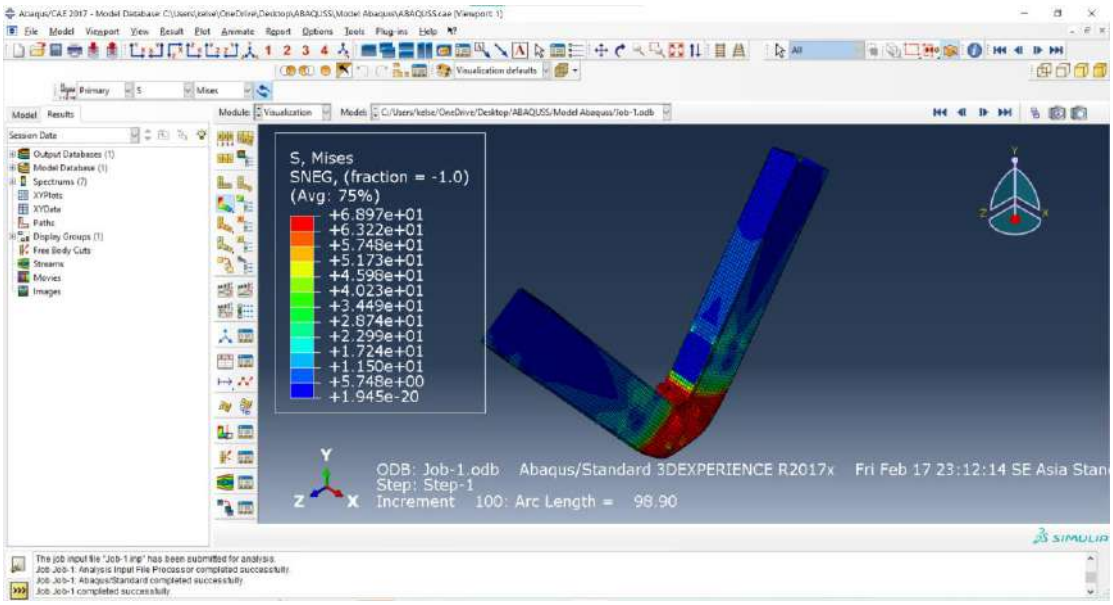

Job Manager
Name
Model
Type
Status
Job-1
Model-1
Full Analysis
Running
Write Input
Monitor...
Results
Kill
Create...
Edit...
Copy...
Rename...
Delete...
Dismiss
RP-1
RP-2
RP-3
RP-4
11484 elements have been generated on part: Part-1
The job "Job-1" has been created.
The job input file "Job-1.inp" has been submitted for analysis.
Job Job-1: Analysis Input File Processor completed successfully.



Setelah *Status Completed*, Pada *Job Manager*, Klik *Results* – Klik *Plot Countours on Deformed Shape.*

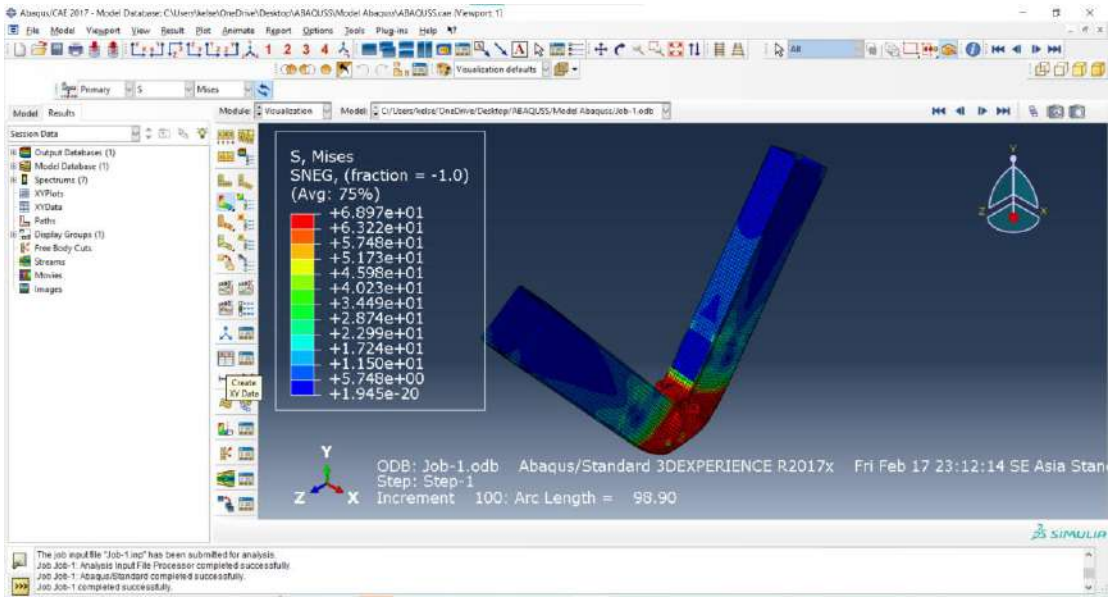


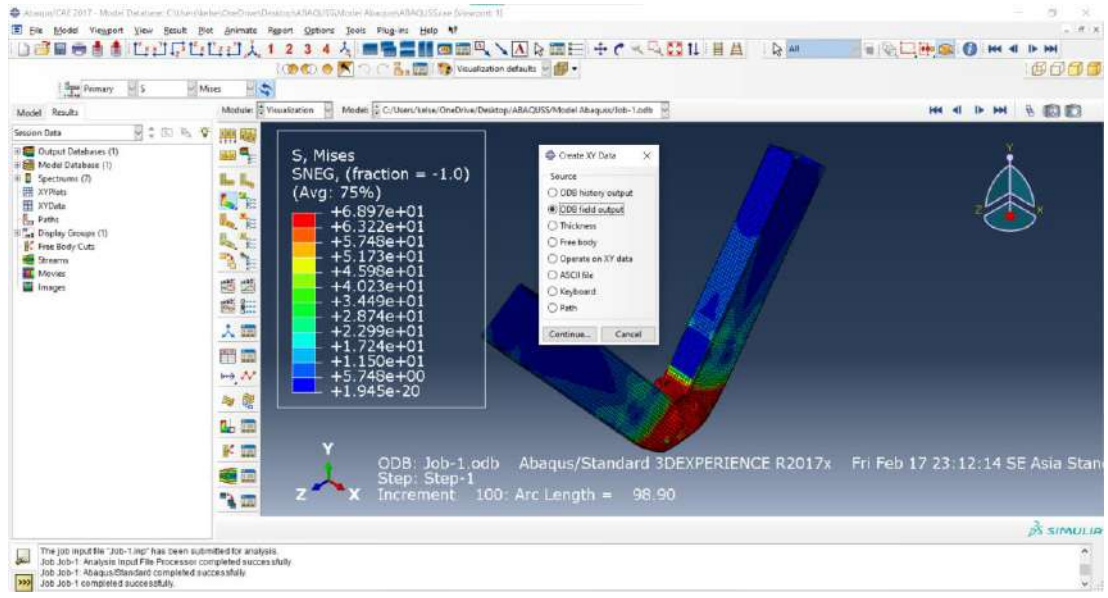


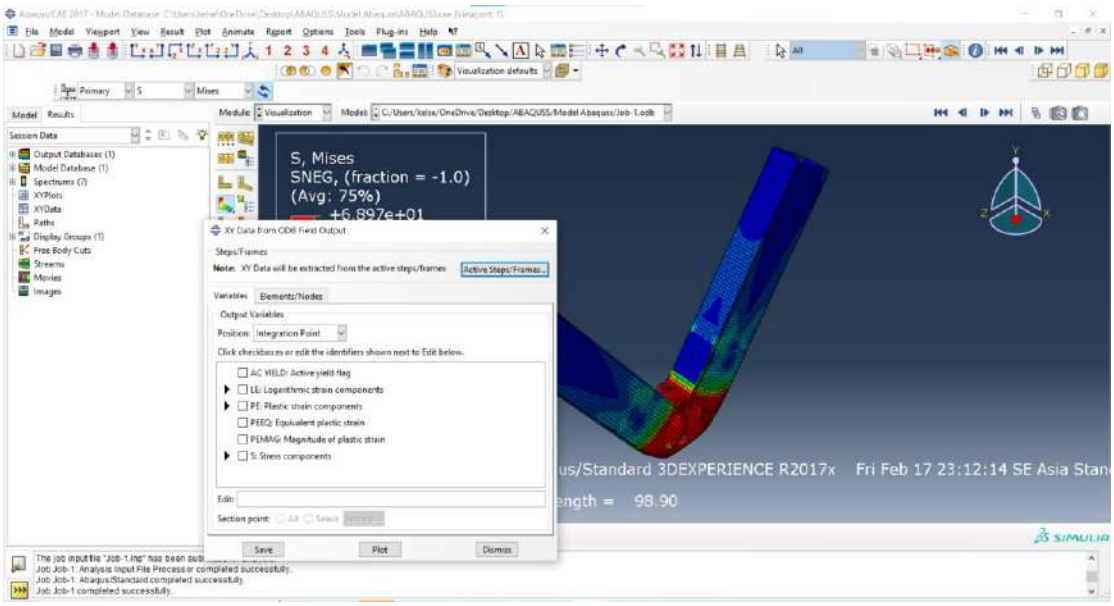


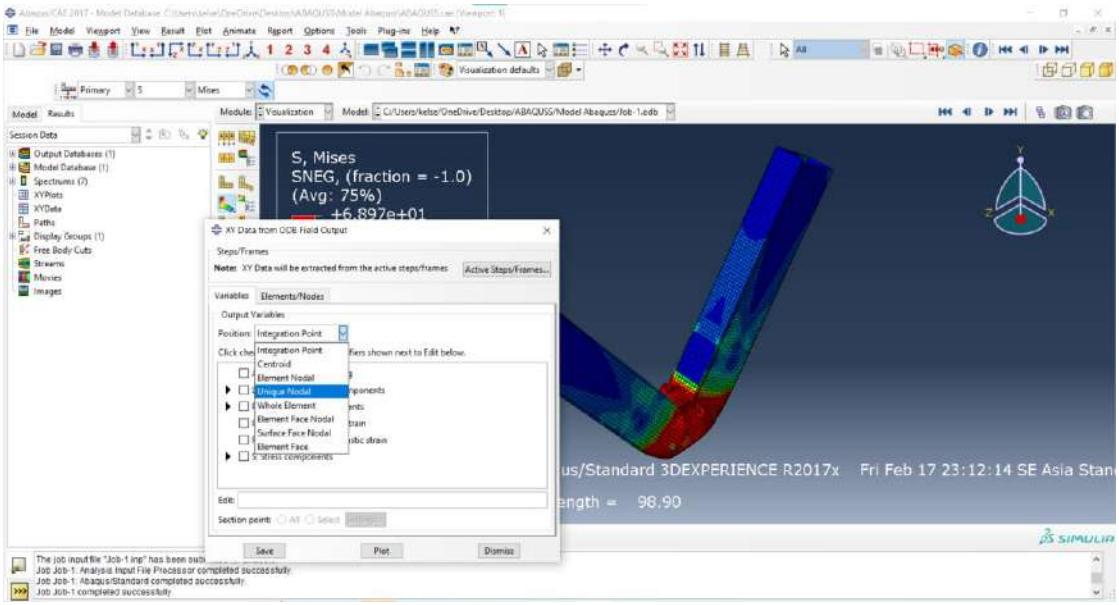
Jika ingin melihat hasil dalam grafik *Force Vs Displacement*:

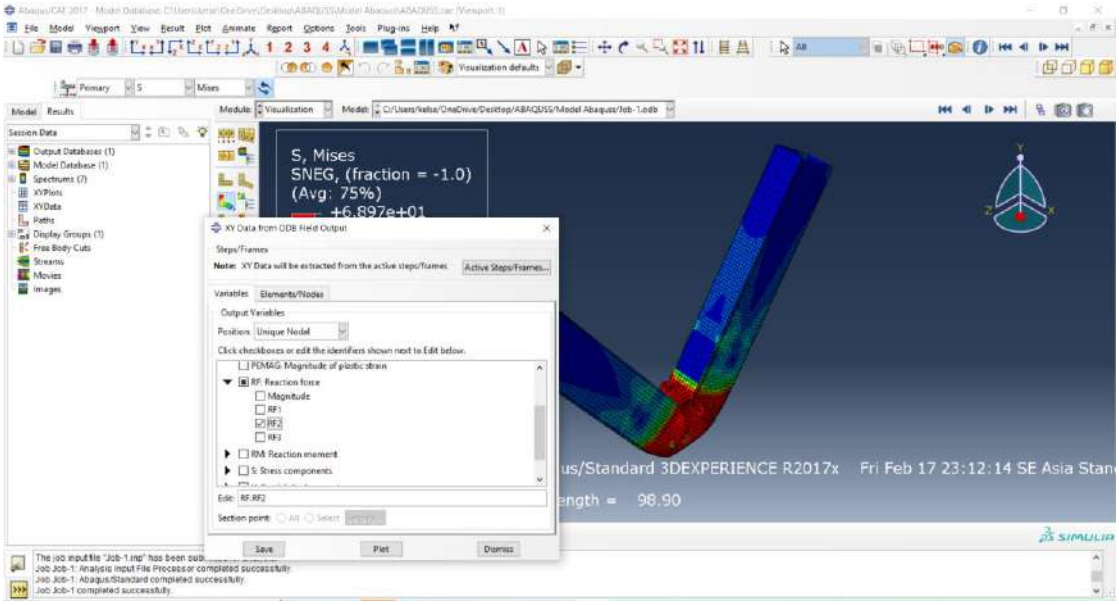
1. Klik *Create XY Data* – Klik *ODB Field Output* – Klik *Continue*… – Pada *XY Data from ODB Field Output*, *Variables – Output Variables* pilih *Unique Nodal* – Klik *RF: Reaction Force* – Centang RF2 – Klik *U: Spatial Displacement* – Centang U2 – Klik *Element/Nodes* – Klik SET-5 – Klik *Plot*.
2. Klik *Create XY Data* – Klik *Operate on XY Data* – Klik *Continue*… – Pada *Operate on XY Data*, ketik seperti digambar – Klik *Plot Expression*.

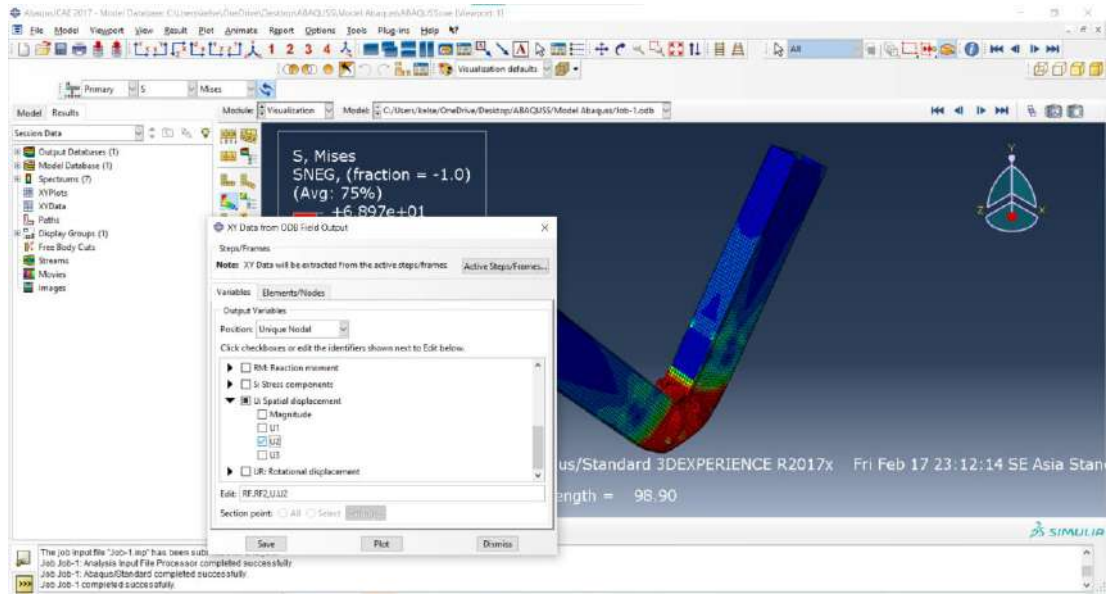

S, Mises
SNEG, (fraction = -1.0)
(Avg: 75%)
+6.897e+01
+6.322e+01
+5.748e+01
+5.173e+01
+4.598e+01
+4.023e+01
+3.449e+01
+2.874e+01
+2.299e+01
+1.724e+01
+1.150e+01
+5.748e+00
+1.945e-20
ODB: Job-1.odb Abaqus/Standard 3DEXPERIENCE R2017x Fri Feb 17 23:12:14 SE Asia Stan
Step: Step-1
Increment 100: Arc Length = 98.90

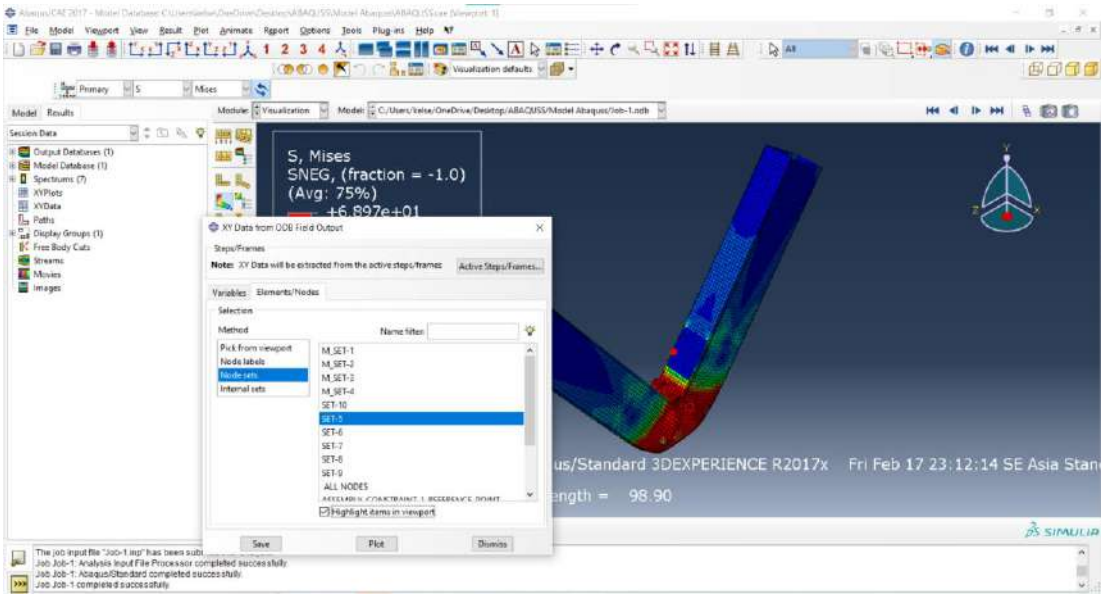

S, Mises
SNEG, (fraction = -1.0)
(Avg: 75%)
+6.897e+01
+6.322e+01
+5.748e+01
+5.173e+01
+4.598e+01
+4.023e+01
+3.449e+01
+2.874e+01
+2.299e+01
+1.724e+01
+1.150e+01
+5.748e+00
+1.945e-20
Create XY Data
Source
ODB history output
ODB field output
Thickness
Free body
Operate on XY data
ASCII file
Keyboard
Path
Continue...
Cancel
ODB: Job-1.odb Abaqus/Standard 3DEXPERIENCE R2017x Fri Feb 17 23:12:14 SE Asia Stan
Step: Step-1
Increment 100: Arc Length = 98.90

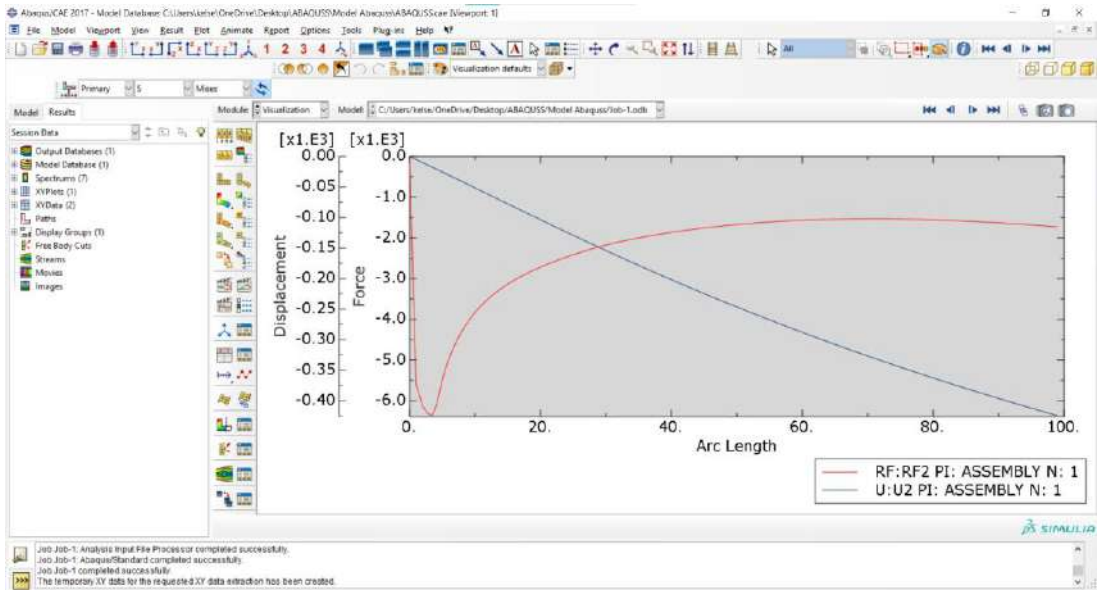

S, Mises
SNEG, (fraction = -1.0)
(Avg: 75%)
XY Data from ODB Field Output
Steps/Frames
Active Steps/Frames...
Variables
Elements/Nodes
Output Variables
Position: Integration Point
Click checkboxes or edit the identifiers shown next to Edit below.
Edit:
Section point:
Save
Plot
Dismiss

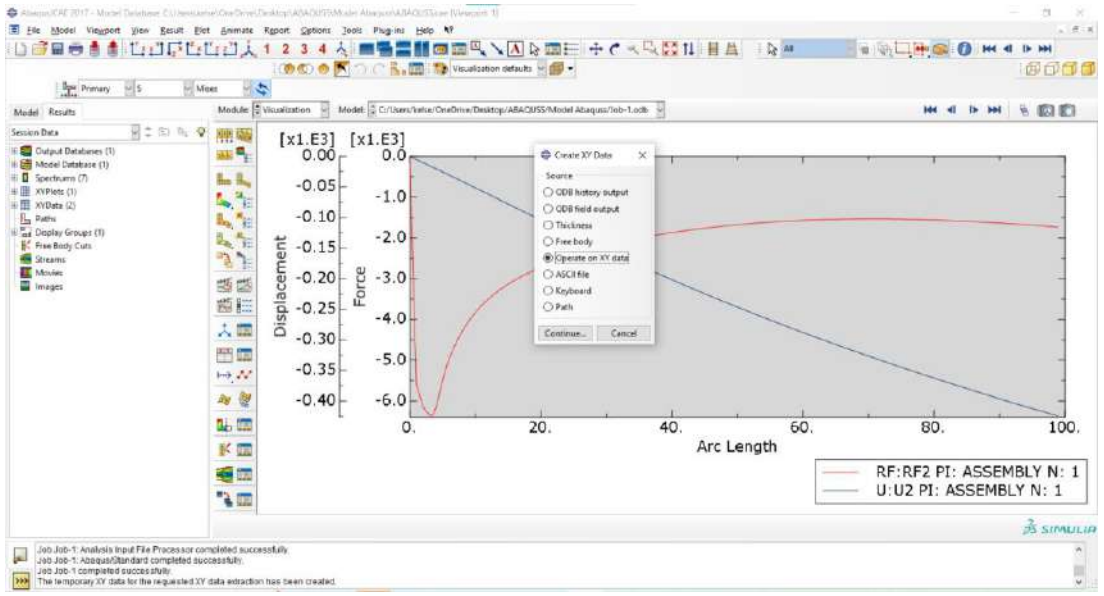

S, Mises
SNEG, (fraction = -1.0)
(Avg: 75%)
+6.897e+01
XY Data from ODB Field Output
Steps/Frames
Note: XY Data will be extracted from the active steps/frames
Active Steps/Frames...
Variables
Elements/Nodes
Output Variables
Position: Integration Point
Integration Point
Centroid
Element Nodal
Unique Nodal
Whole Element
Element Face Nodal
Surface Face Nodal
Element Face
Edit:
Section point:
Save
Plot
Dismiss
us/Standard 3DEXPERIENCE R2017x
Fri Feb 17 23:12:14 SE Asia Stan
ength = 98.90
SIMULIA
The job input file "Job-1.inp" has been sub
Job Job-1: Analysis Input File Processor completed successfully.
Job Job-1: Abaqus/Standard completed successfully.
Job Job-1 completed successfully.

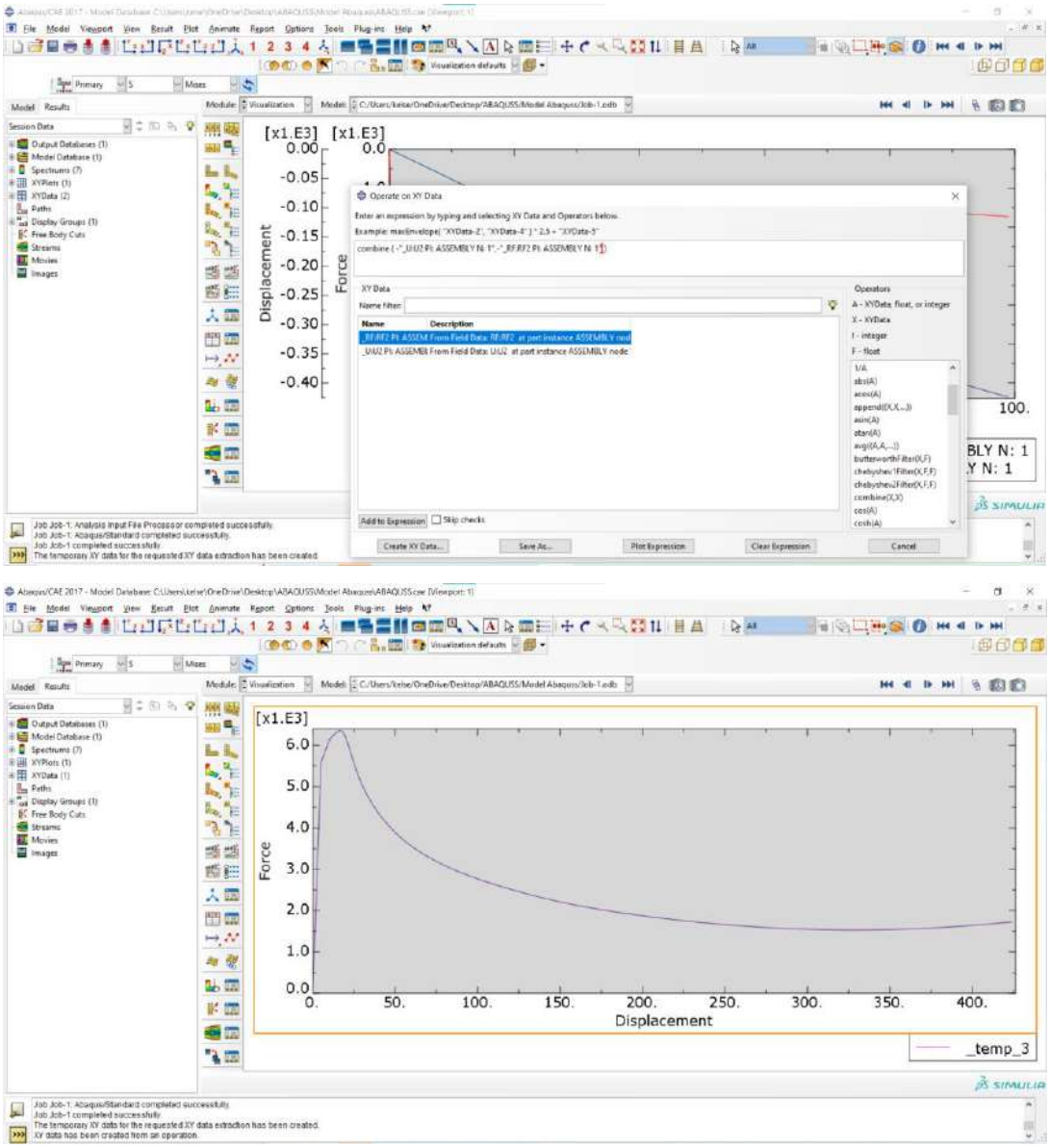

S, Mises
SNEG, (fraction = -1.0)
(Avg: 75%)
+6.897e+01
XY Data from ODB Field Output
Steps/Frames
Note: XY Data will be extracted from the active steps/frames
Active Steps/Frames...
Variables
Elements/Nodes
Output Variables
Position: Unique Nodal
Click checkboxes or edit the identifiers shown next to Edit below.
PEMAG: Magnitude of plastic strain
RF: Reaction force
Magnitude
RF1
RF2
RF3
RM: Reaction moment
S: Stress components
Edit: RF.RF2
Section point:
Save
Plot
Dismiss
us/Standard 3DEXPERIENCE R2017x
Fri Feb 17 23:12:14 SE Asia Stan
ength = 98.90
SIMULIA
The job input file "Job-1.inp" has been sub
Job Job-1: Analysis Input File Processor completed successfully.
Job Job-1: Abaqus/Standard completed successfully.
Job Job-1 completed successfully.


S, Mises
SNEG, (fraction = -1.0)
(Avg: 75%)
+6.897e+01
XY Data from ODB Field Output
Steps/Frames
Note: XY Data will be extracted from the active steps/frames
Active Steps/Frames...
Variables
Elements/Nodes
Output Variables
Position: Unique Nodal
Click checkboxes or edit the identifiers shown next to Edit below.
RM: Reaction moment
S: Stress components
U: Spatial displacement
Magnitude
U1
U2
U3
UR: Rotational displacement
Edit: RF.RF2,U.U2
Section point:
Save
Plot
Dismiss
us/Standard 3DEXPERIENCE R2017x
Fri Feb 17 23:12:14 SE Asia Stan
ength = 98.90
SIMULIA
The job input file "Job-1.inp" has been sub
Job Job-1: Analysis Input File Processor completed successfully.
Job Job-1: Abaqus/Standard completed successfully.
Job Job-1 completed successfully.

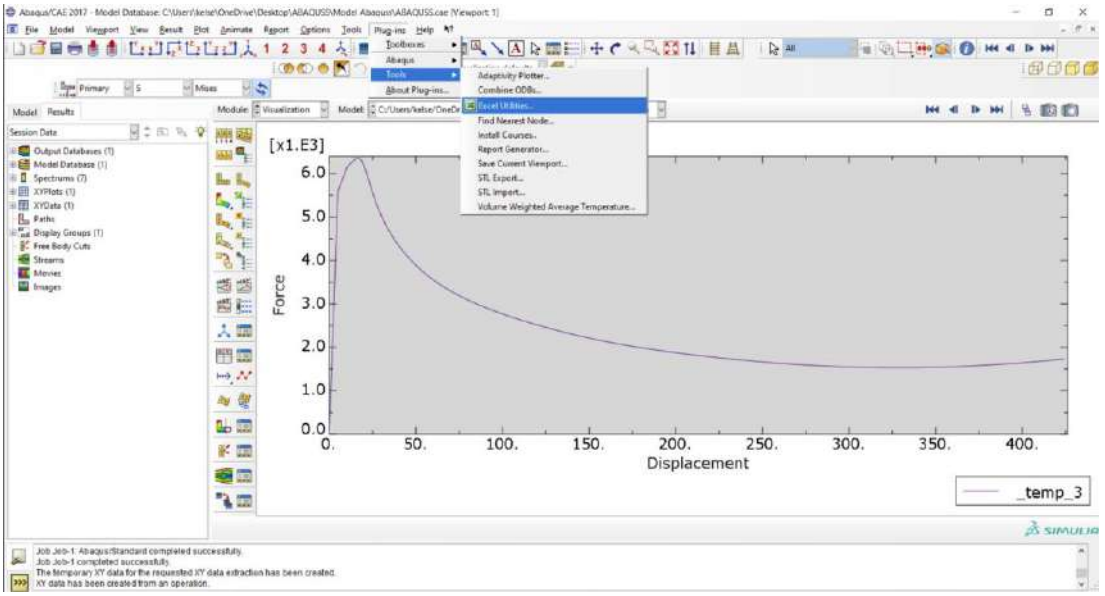

S, Mises
SNEG, (fraction = -1.0)
(Avg: 75%)
XY Data from ODB Field Output
Steps/Frames
Variables
Elements/Nodes
Selection
Method
Pick from viewport
Node labels
Node sets
Internal sets
Name filter:
M_SET-1
M_SET-2
M_SET-3
M_SET-4
SET-10
SET-5
SET-6
SET-7
SET-8
SET-9
ALL NODES
Save
Plot
Dismiss
Fri Feb 17 23:12:14 SE Asia Stan
98.90

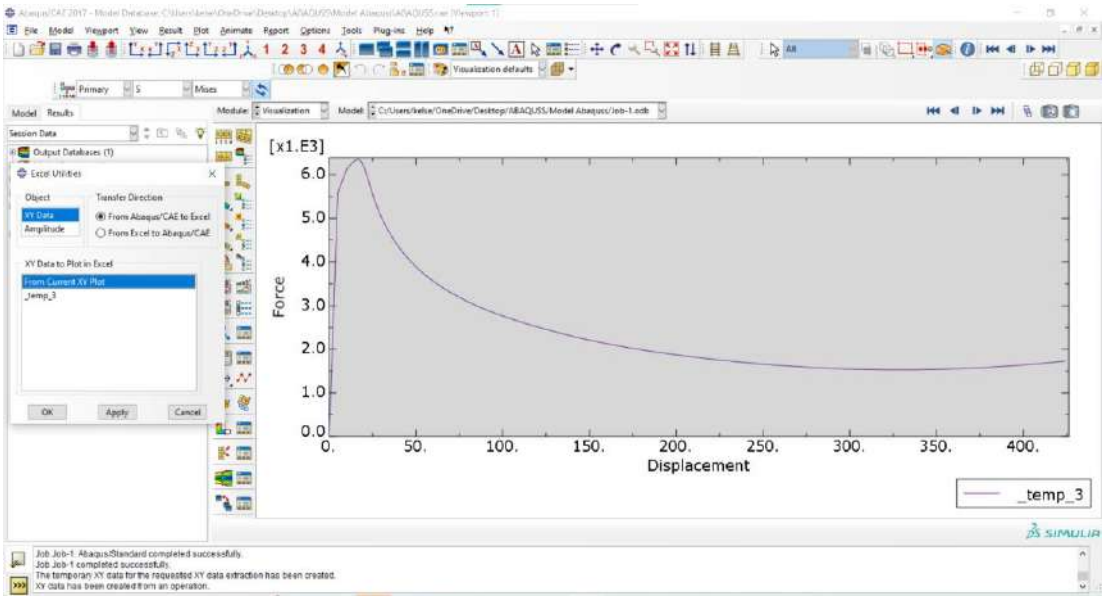

[x1.E3]
[x1.E3]
Displacement
Force
Arc Length
RF:RF2 PI: ASSEMBLY N: 1
U:U2 PI: ASSEMBLY N: 1

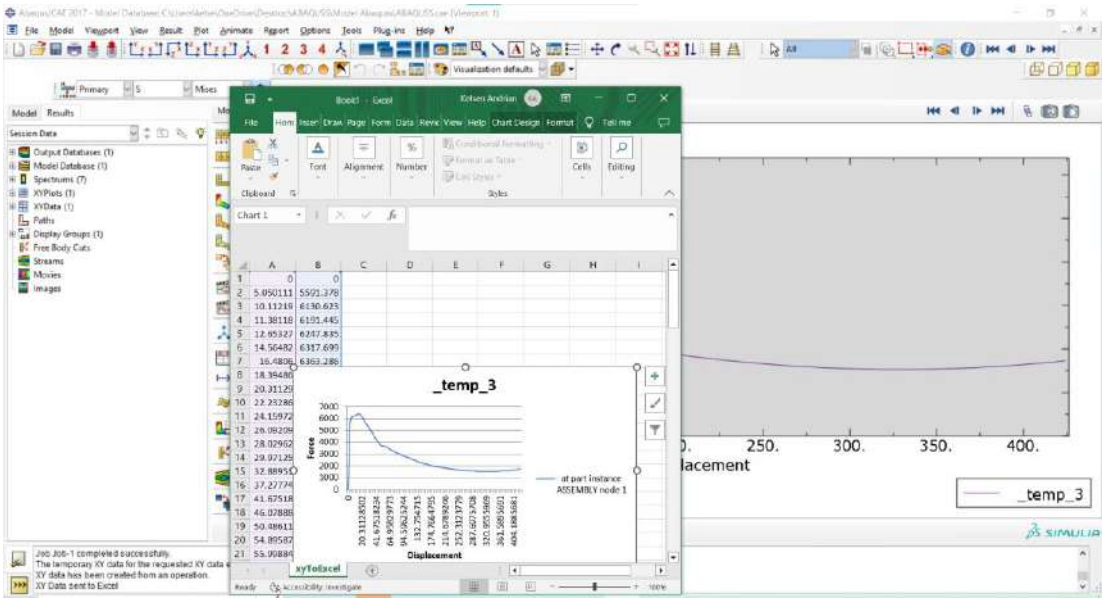

[x1.E3]
[x1.E3]
Create XY Data
Source
ODB history output
ODB field output
Thickness
Free body
Operate on XY data
ASCII file
Keyboard
Path
Continue...
Cancel
Displacement
Force
Arc Length
RF:RF2 PI: ASSEMBLY N: 1
U:U2 PI: ASSEMBLY N: 1



Jika ingin melihat data dalam angka tabel excel, Klik *Plug-ins* – Klik *Tools* – Klik *Excel Utilities…* – Pada *Excel Utilities*, Klik *From Current XY Plot* – Klik Ok.


[x1.E3]
Force
Displacement
_temp_3


Excel Utilities
Object
XY Data
Amplitude
Transfer Direction
From Abaqus/CAE to Excel
From Excel to Abaqus/CAE
XY Data to Plot in Excel
From Current XY Plot
_temp_3
OK
Apply
Cancel
[x1.E3]
Force
Displacement
_temp_3


_temp_3
xyToExcel
Displacement
_temp_3

DAFTAR PUSTAKA

ABAQUS. (2023). User's manual and theory manual.: Dassault Systèmes Simulia Corp.

Chen, Z., Huang, Y., & Young, B. (2022). Design of Cold-formed Ferritic Stainless Steel RHS Perforated Beams. *Engineering Structures, 250*.

Huang, Y., & Young, B. (2014). Stress-strain Relationship of Cold-formed Lean Duplex Stainless Steel at Elevated Temperatures. *Journal of Constructional Steel Research, 92*, 103-113.