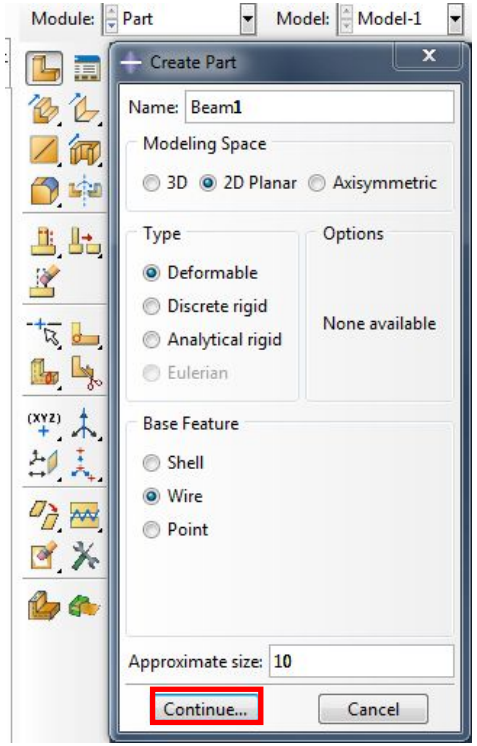
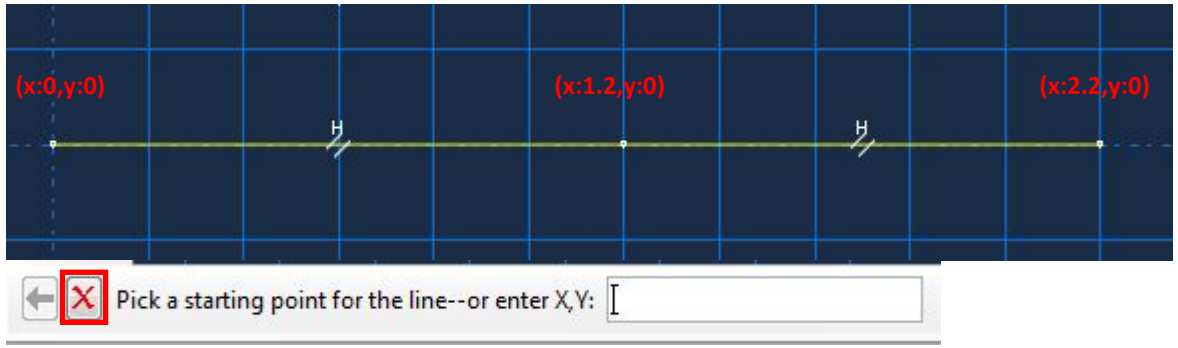
Моделирование многопролетной балки с узловым шарниром

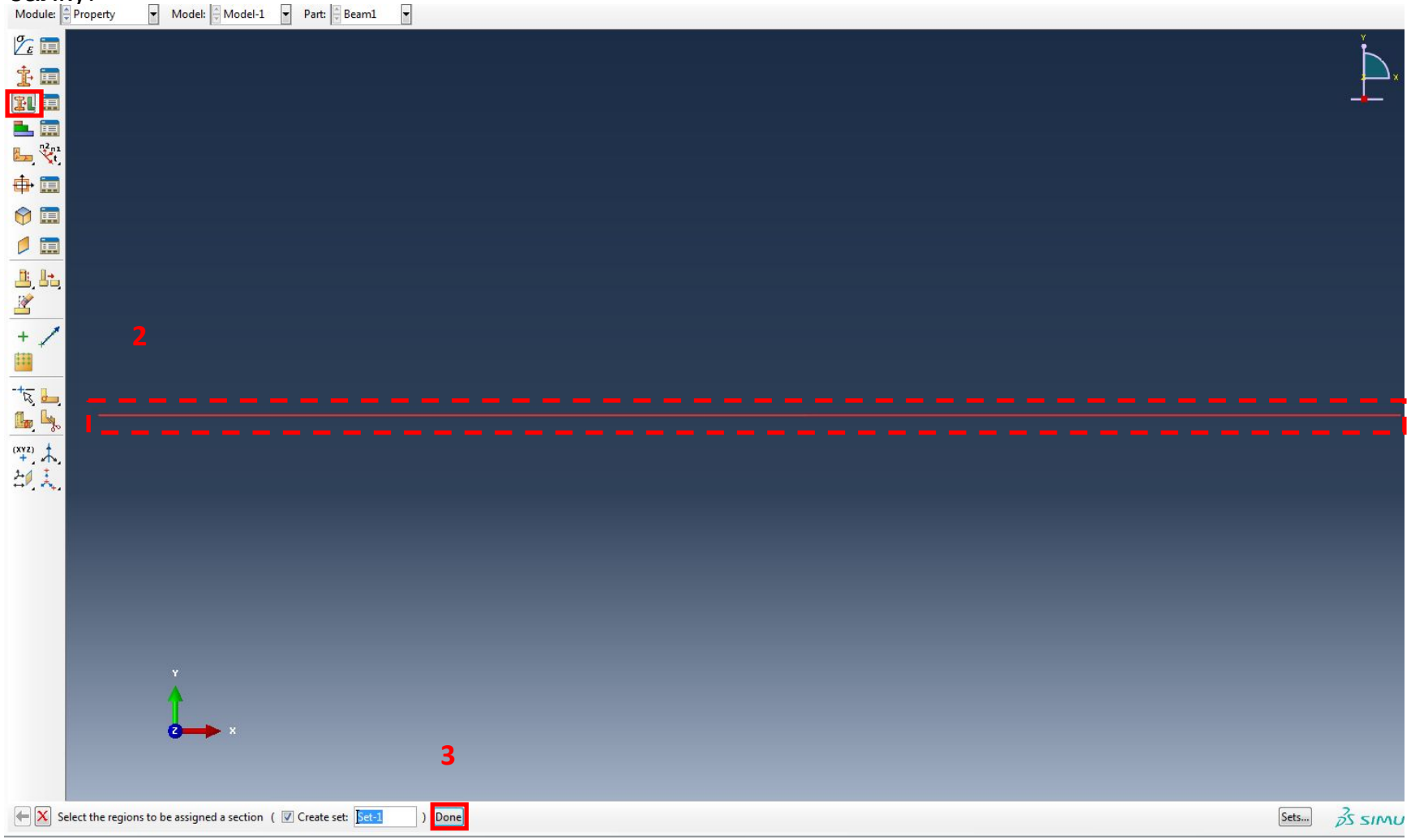
Схема балки и нагружения:

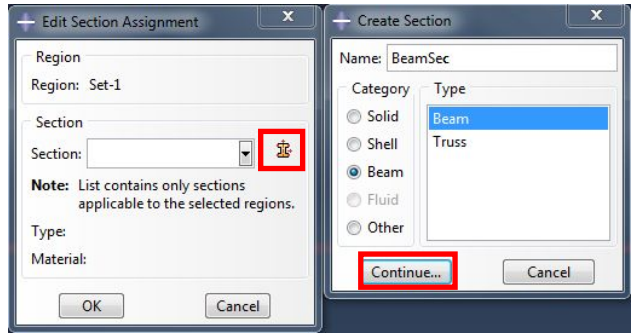
Шаг 1. Создадим новый Part в виде балки Beam1. Для этого в модуле Part применим инструмент Create Part:

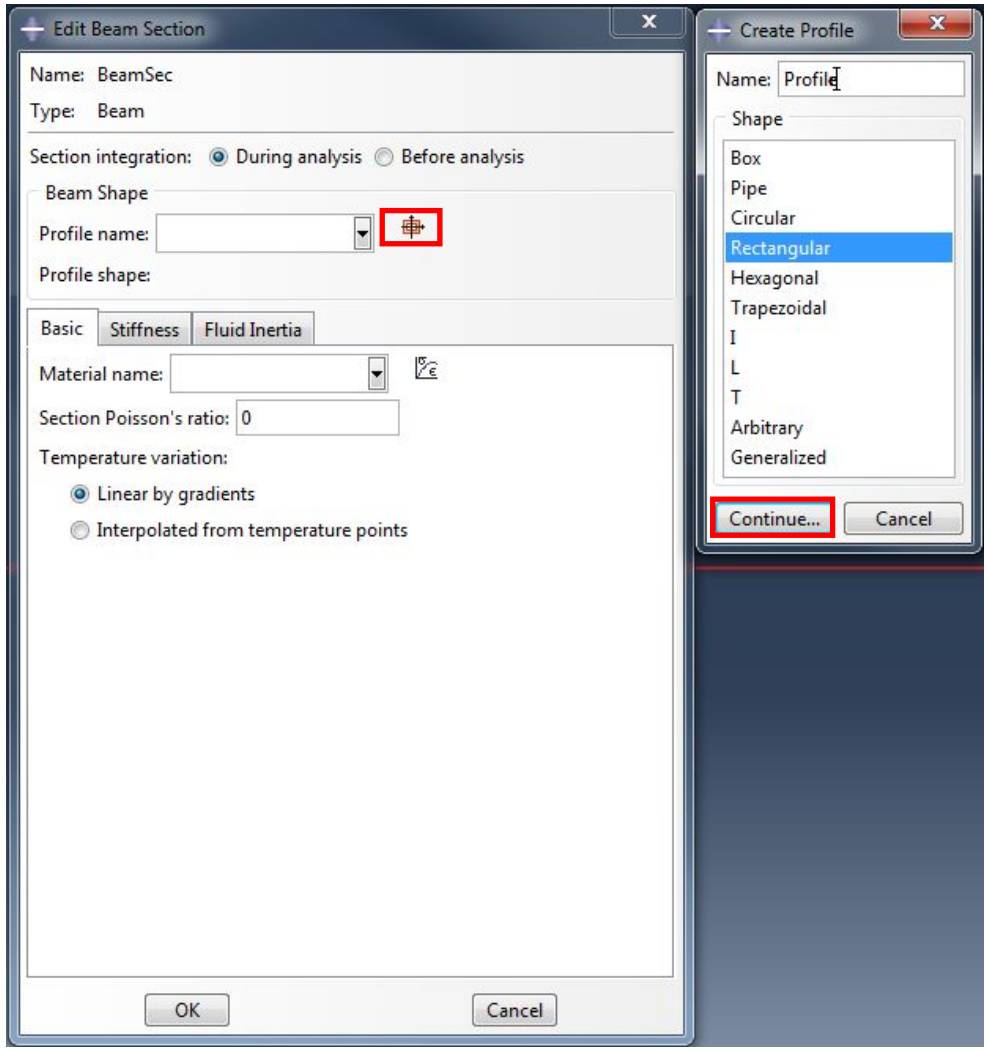


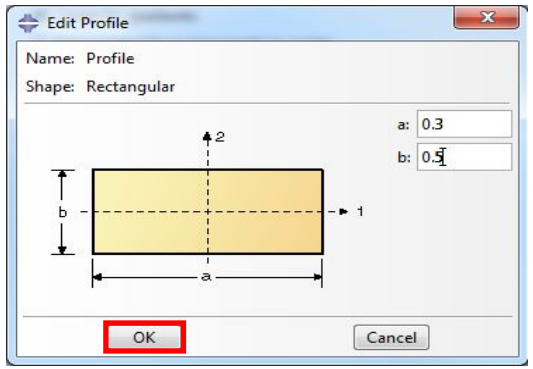


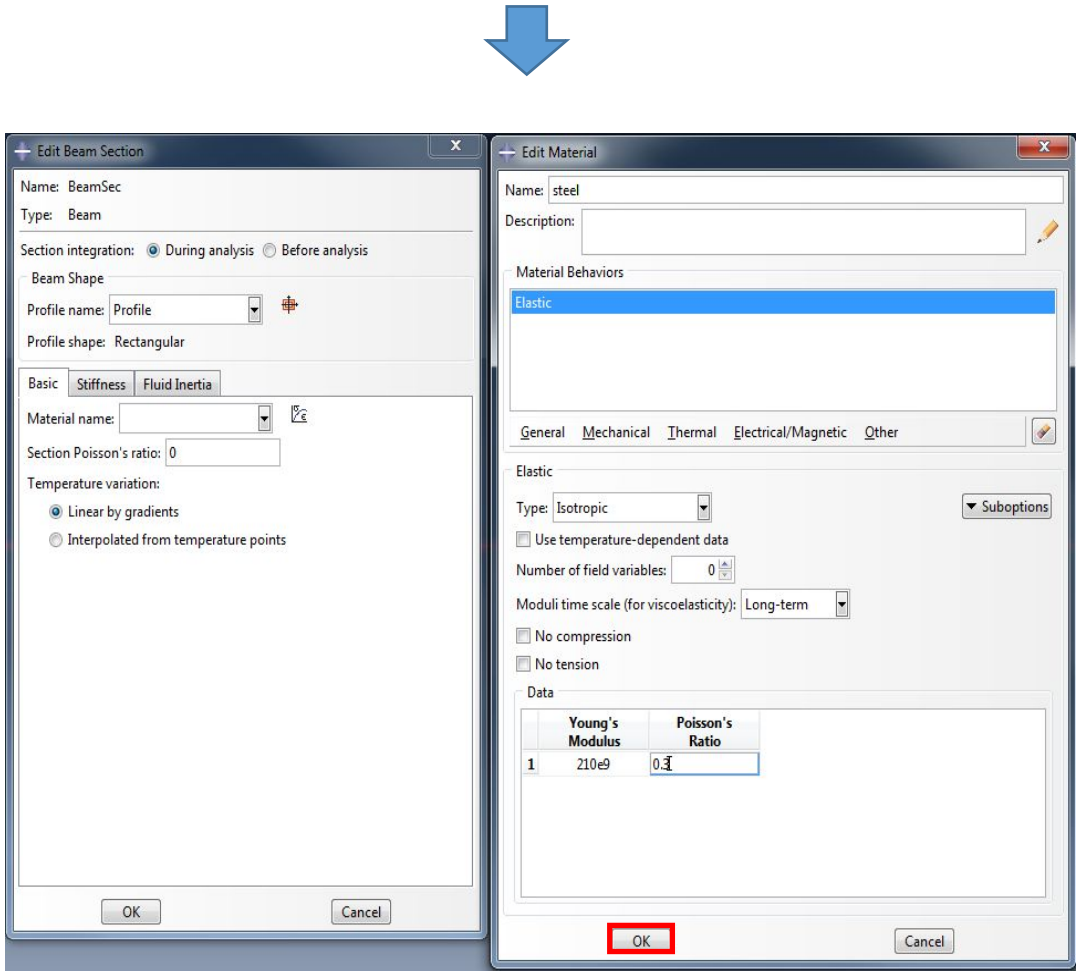
Шаг 2. Назначим для балки сечение. Для этого в модуле Property выберем инструмент Assign Section и выберем нашу балку:

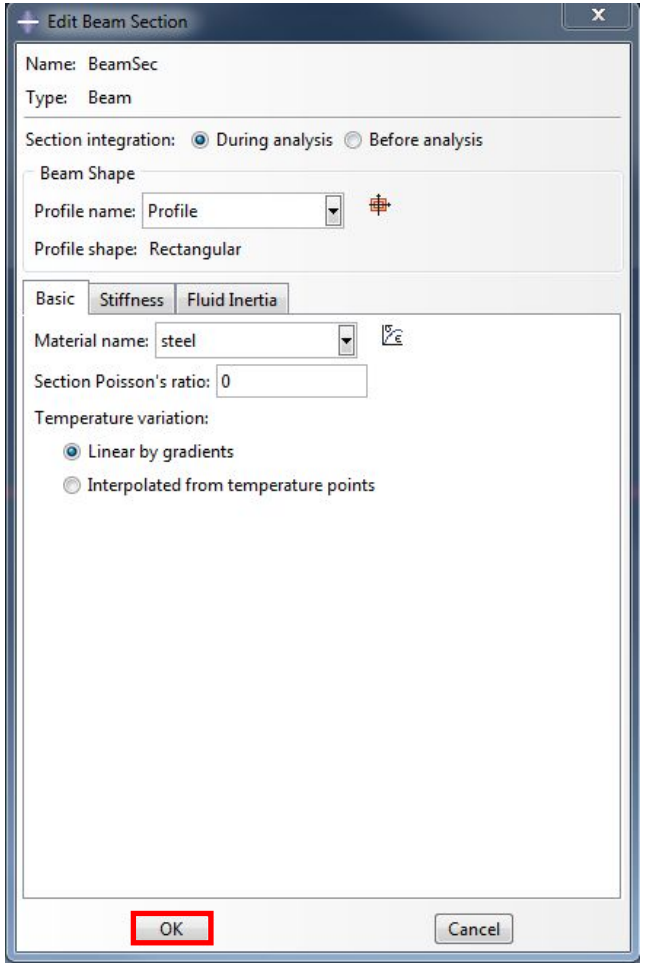


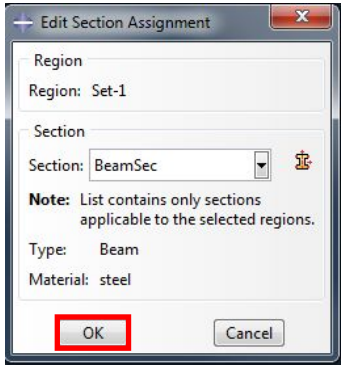



Edit Beam Section
Name: BeamSec
Type: Beam
Section integration: During analysis Before analysis
Beam Shape
Profile name:
Profile shape:
Basic
Stiffness
Fluid Inertia
Material name:
Section Poisson's ratio: 0
Temperature variation:
Linear by gradients
Interpolated from temperature points
OK
Cancel
Create Profile
Name: Profile
Shape
Box
Pipe
Circular
Rectangular
Hexagonal
Trapezoidal
I
L
T
Arbitrary
Generalized
Continue...
Cancel


Edit Profile
Name: Profile
Shape: Rectangular
a: 0.3
b: 0.5
2
1
b
a
OK
Cancel

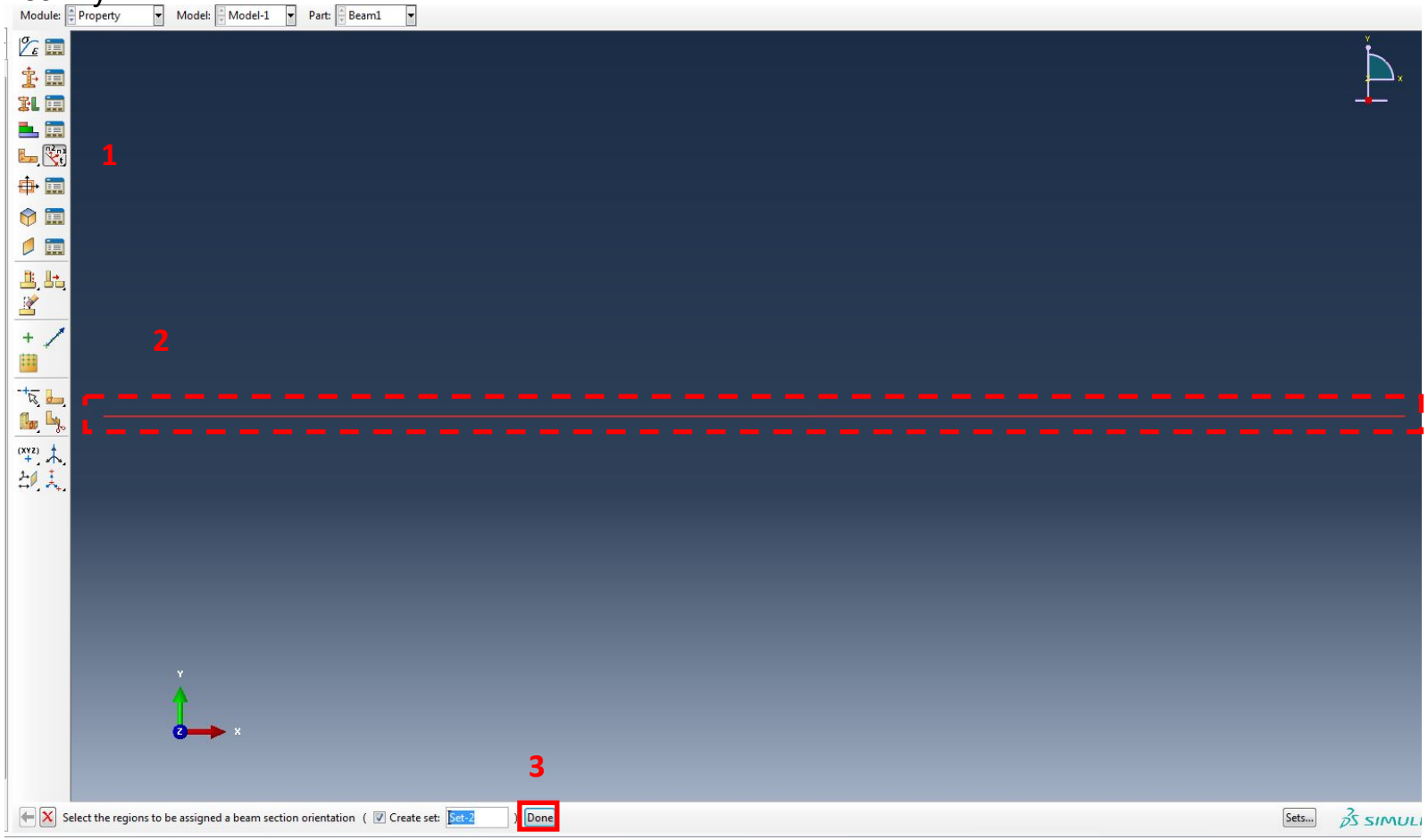

Edit Beam Section
Name: BeamSec
Type: Beam
Section integration: During analysis Before analysis
Beam Shape
Profile name: Profile
Profile shape: Rectangular
Basic Stiffness Fluid Inertia
Material name:
Section Poisson's ratio: 0
Temperature variation:
Linear by gradients
Interpolated from temperature points
OK Cancel
Edit Material
Name: steel
Description:
Material Behaviors
Elastic
General Mechanical Thermal Electrical/Magnetic Other
Elastic
Type: Isotropic
Suboptions
Use temperature-dependent data
Number of field variables: 0
Moduli time scale (for viscoelasticity): Long-term
No compression
No tension
Data
Young's Modulus
Poisson's Ratio
1 210e9 0.3
OK Cancel


Edit Beam Section
Name: BeamSec
Type: Beam
Section integration: During analysis Before analysis
Beam Shape
Profile name: Profile
Profile shape: Rectangular
Basic Stiffness Fluid Inertia
Material name: steel
Section Poisson's ratio: 0
Temperature variation:
Linear by gradients
Interpolated from temperature points
OK Cancel

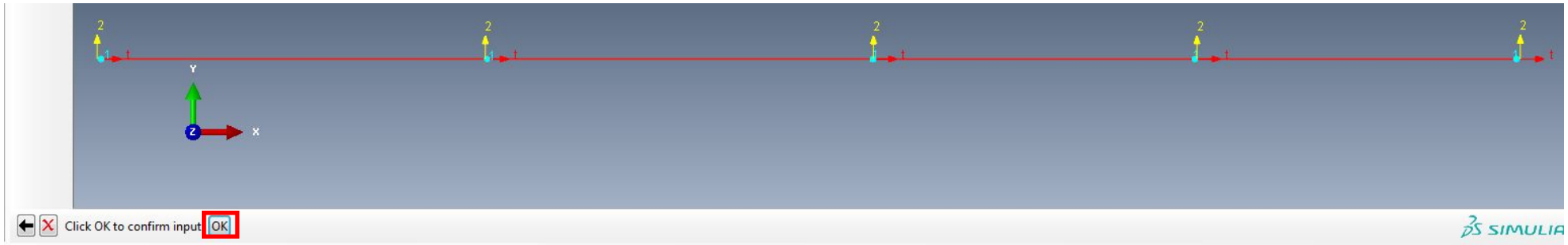

Edit Section Assignment
Region
Region: Set-1
Section
Section: BeamSec
Note: List contains only sections applicable to the selected regions.
Type: Beam
Material: steel
OK Cancel

Select the regions to be assigned a section (Create set: Set-2) Done

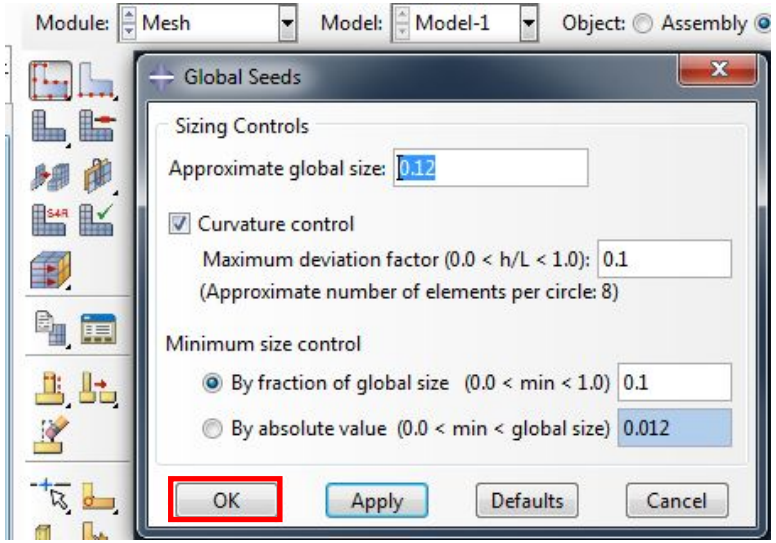
Шаг 3. Назначим ориентацию балки. Для этого в модуле Property выберем инструмент Assign Beam Orientation и выберем нашу балку:

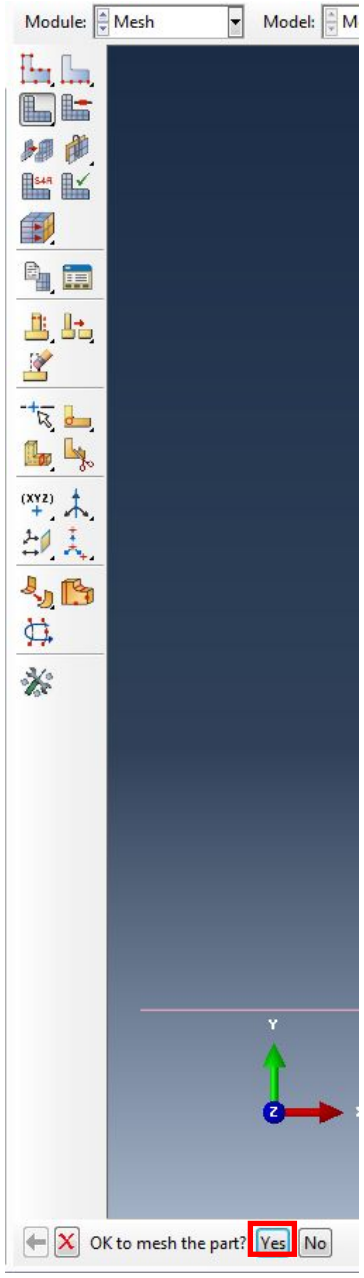


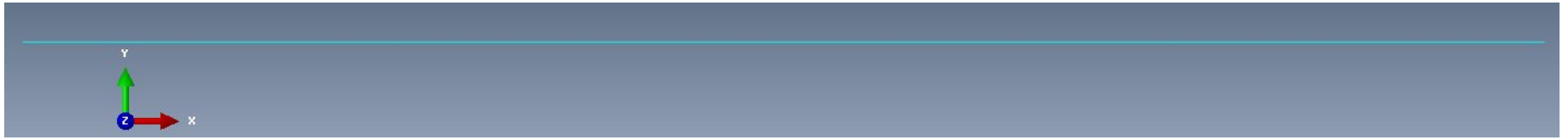
ENTER



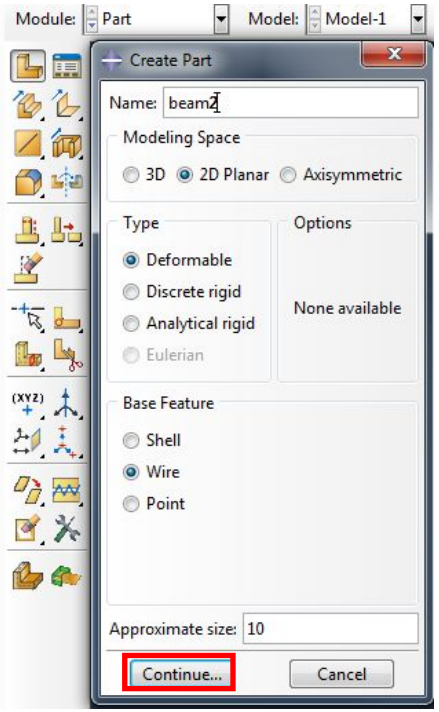
Шаг 4. Создадим КЭ сетку для нашей модели балки. Для этого вначале зададим величину разбиения, выбрав инструмент Seed Part и введем следующие
параметры:

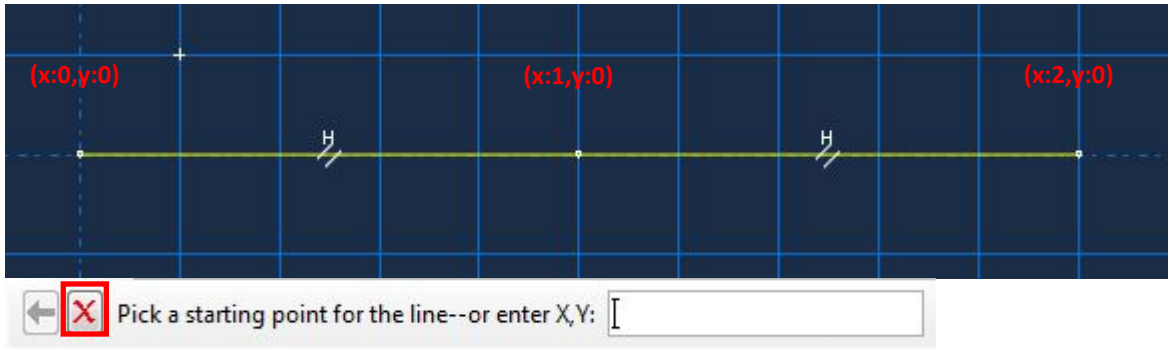



Module: Mesh
Model: M
Y
Z
OK to mesh the part?
Yes
No

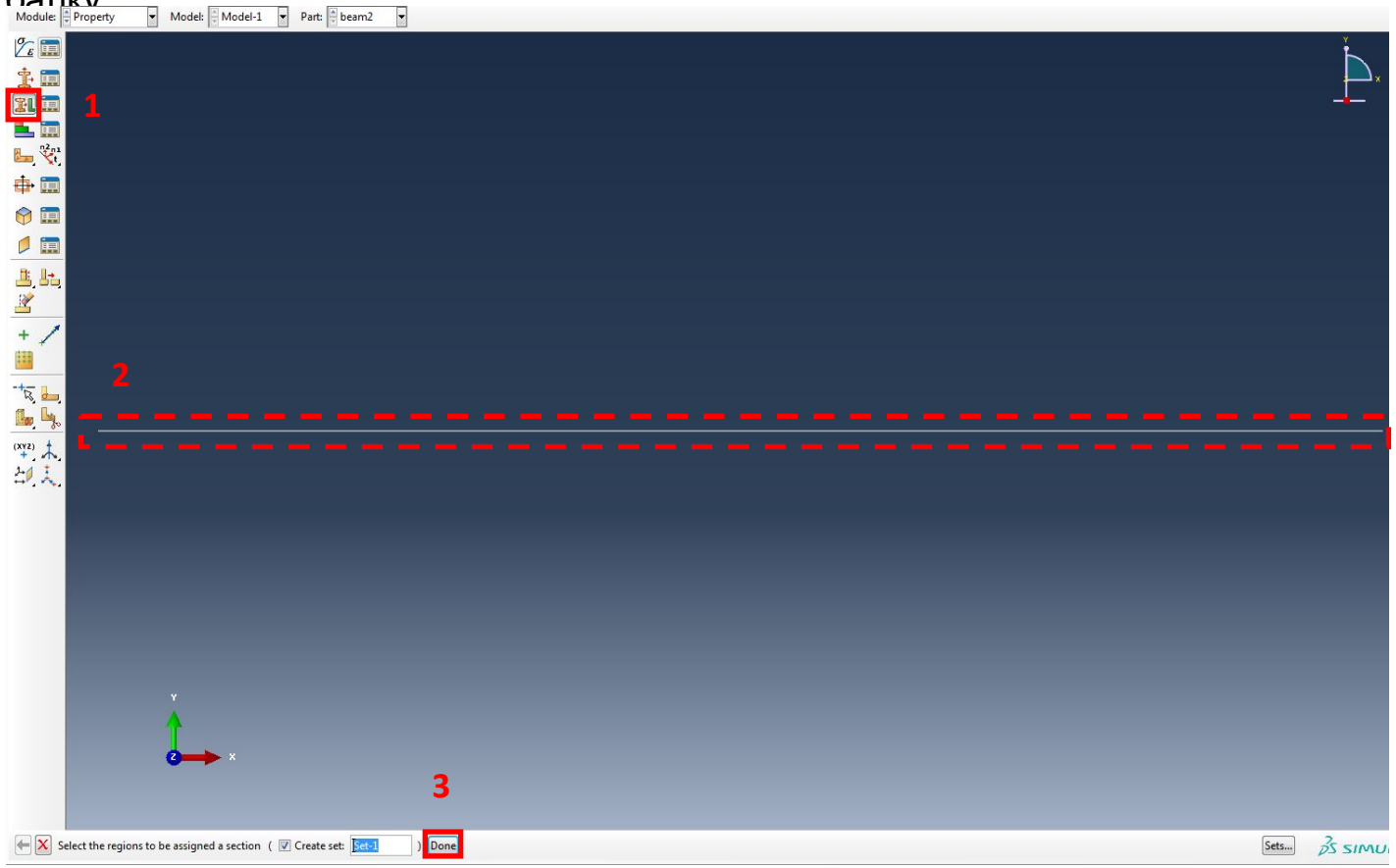

Y
Z
X

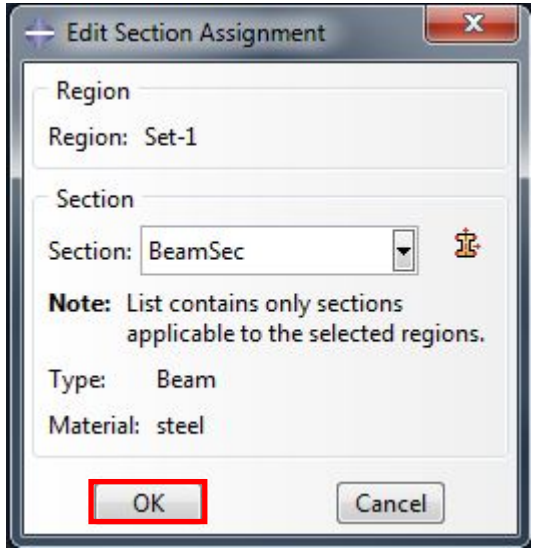
Шаг 5. Создадим новый Part в виде балки Beam2. Для этого в модуле Part применим инструмент Create Part:



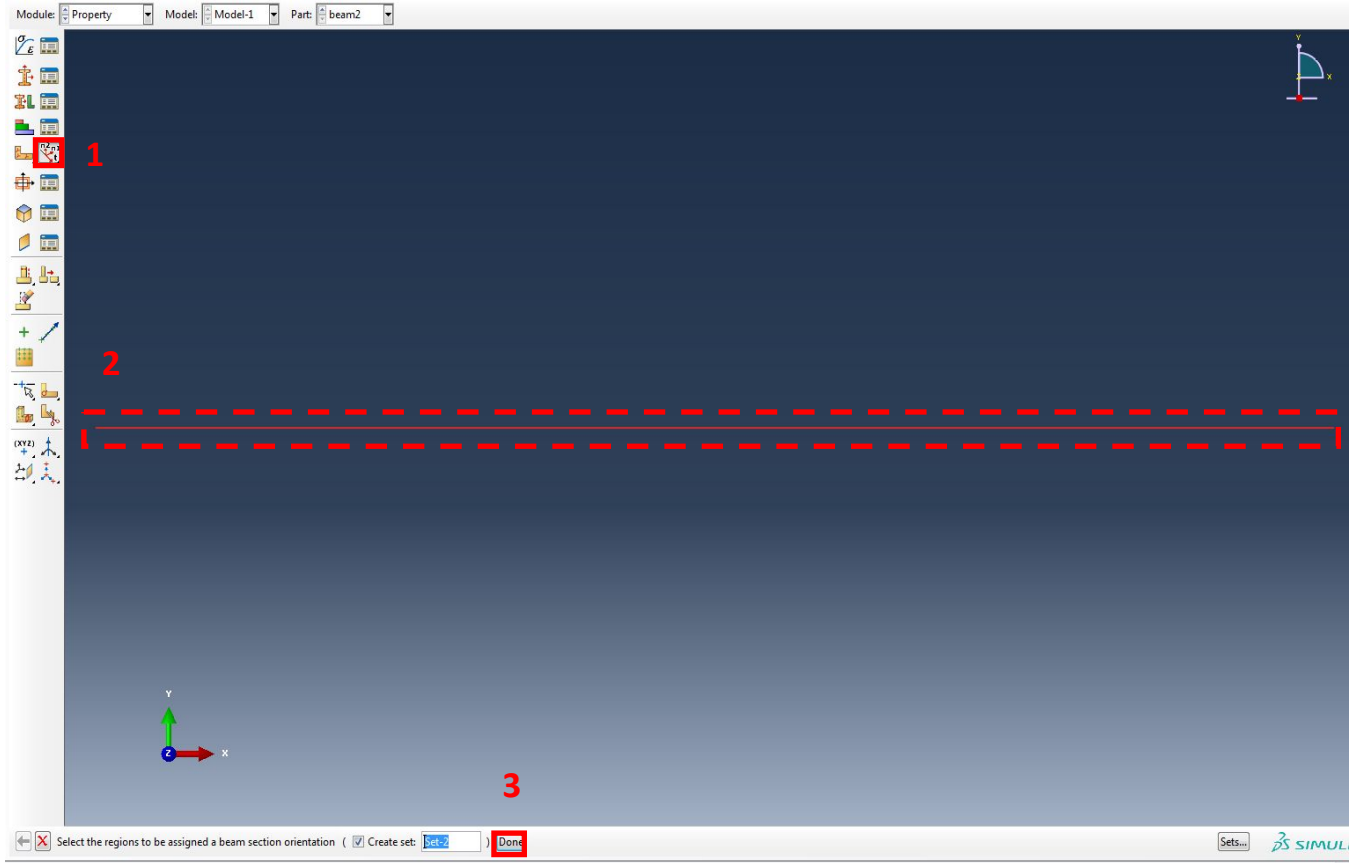


Шаг 6. Назначим для балки сечение. Для этого в модуле Property выберем инструмент Assign Section и выберем нашу балку:

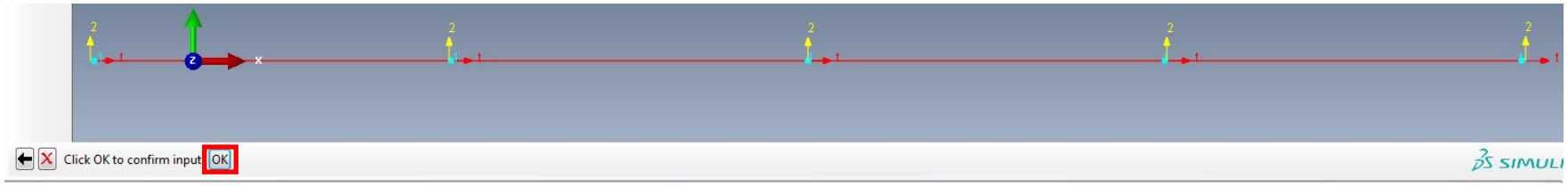




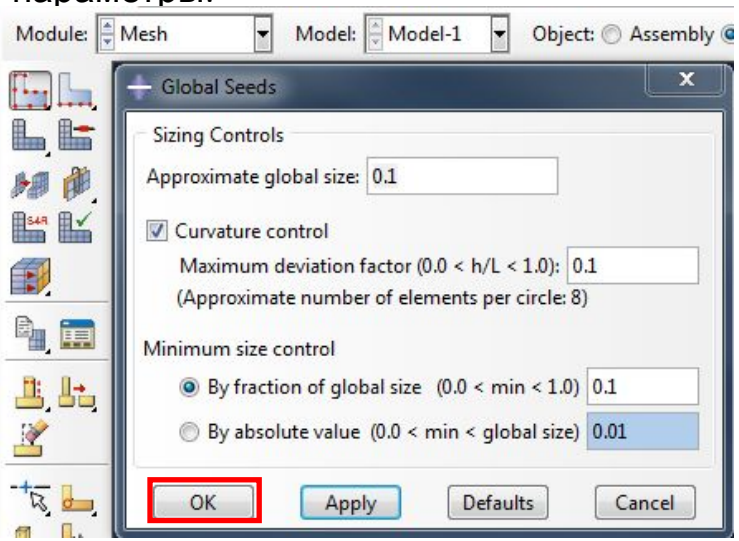
Шаг 7. Назначим ориентацию балки. Для этого в модуле Property выберем инструмент Assign Beam Orientation и выберем нашу балку:

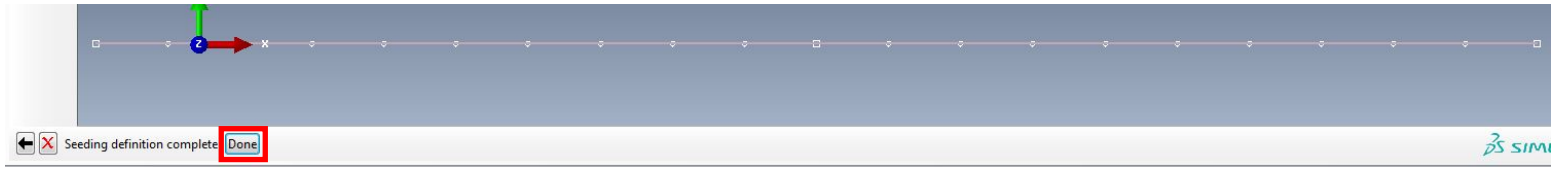


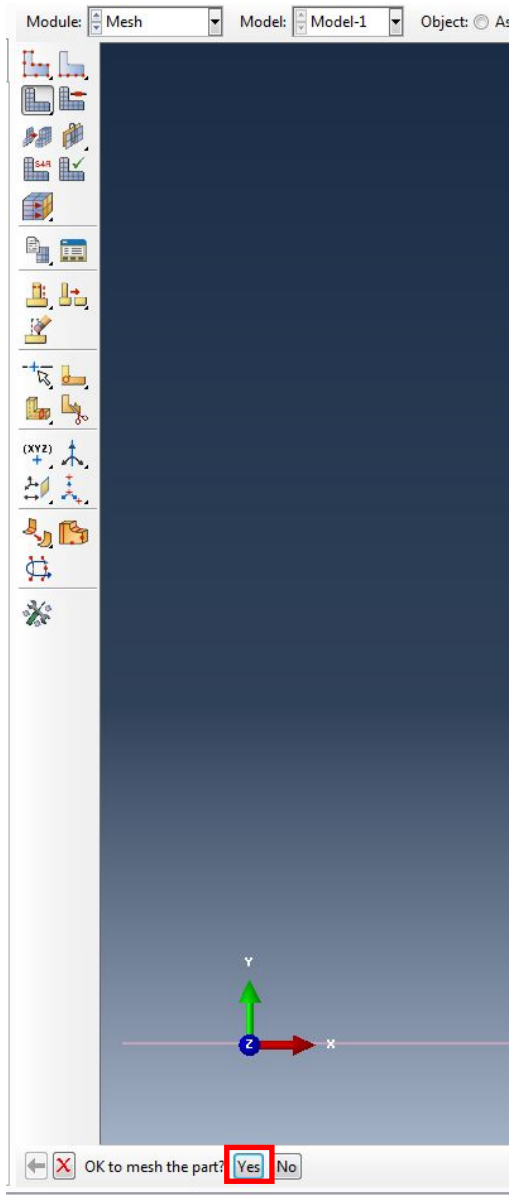
ENTER

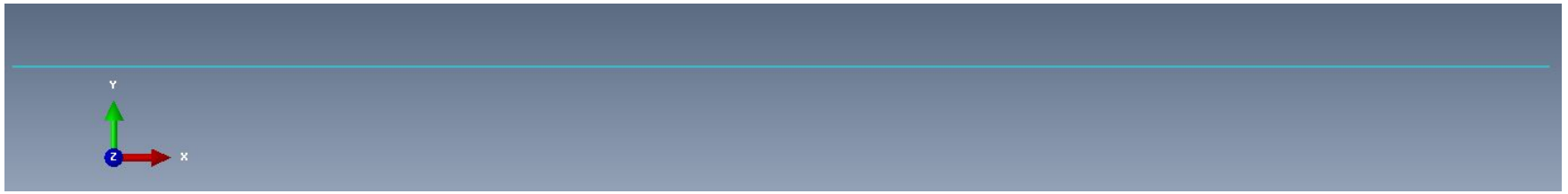


Шаг 8. Создадим КЭ сетку для нашей модели балки. Для этого вначале зададим величину разбиения, выбрав инструмент Seed Part и введем следующие
параметры:

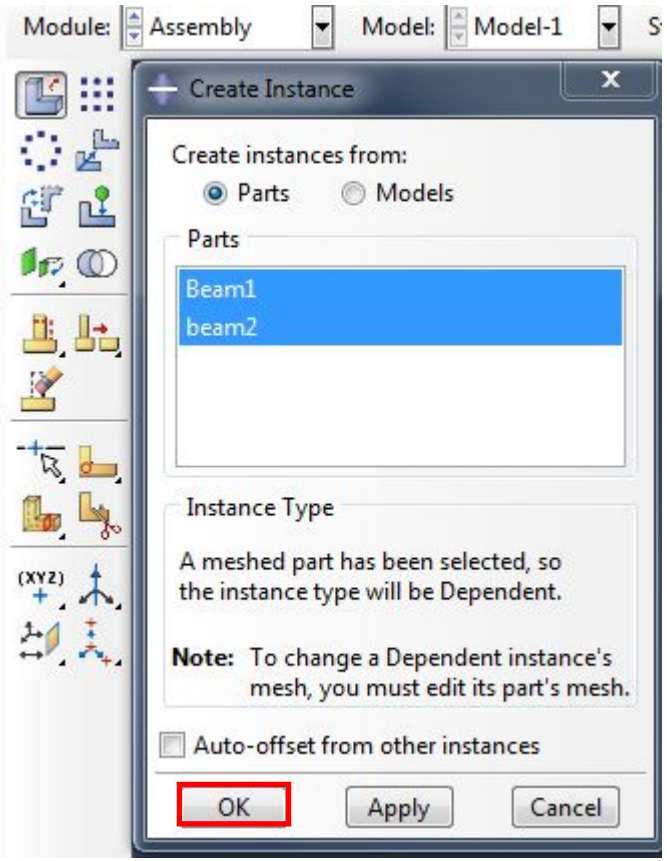




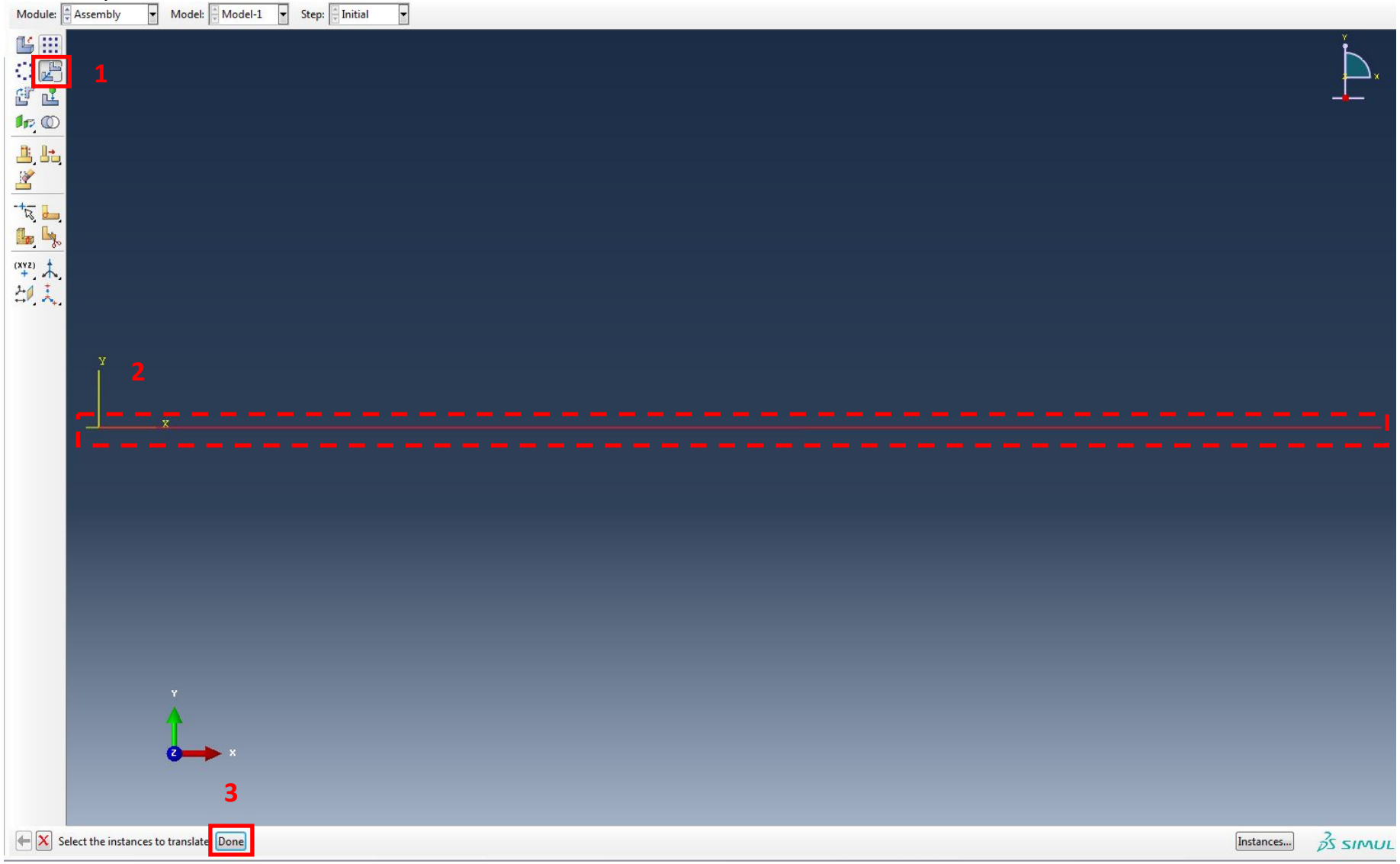

Module: Mesh
Model: Model-1
Object:
OK to mesh the part?
Yes
No



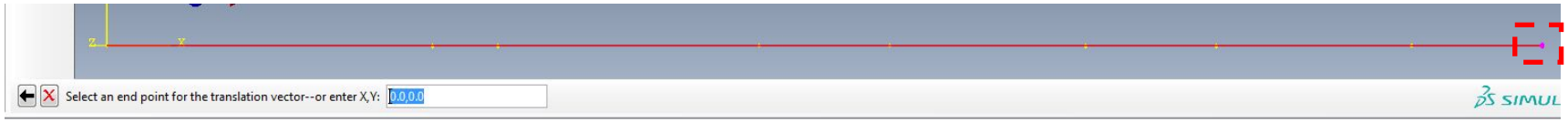
Шаг 9. Соберем нашу модель. Для этого в модуле ASSEMBLY выберем инструмент Create Instance:



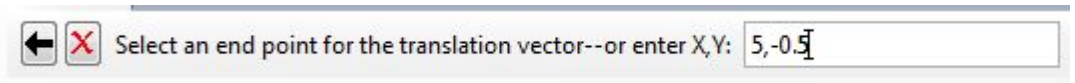
Шаг 10. Далее разнесем наши балки, выбрав инструмент Translate Instance и нашу балку (большей



Выберем в качестве начальной точки крайне правый узел нашей балки:

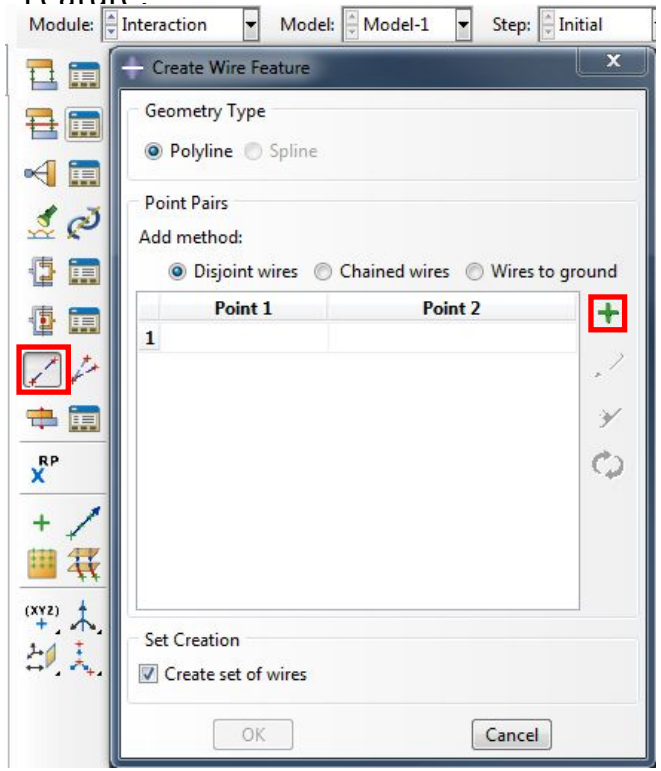


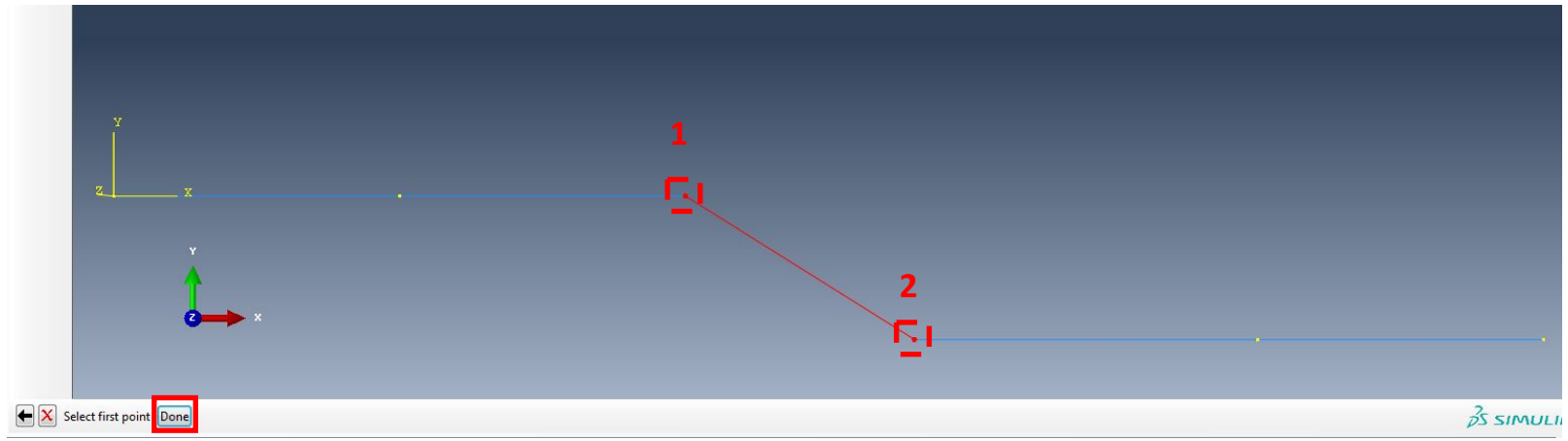
В качестве конечной точки введем вектор:

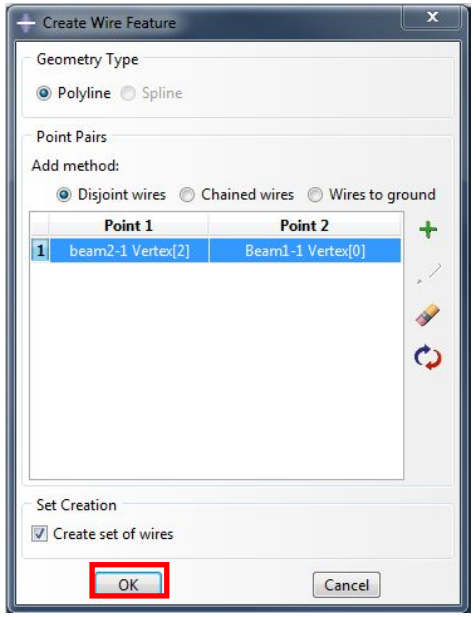


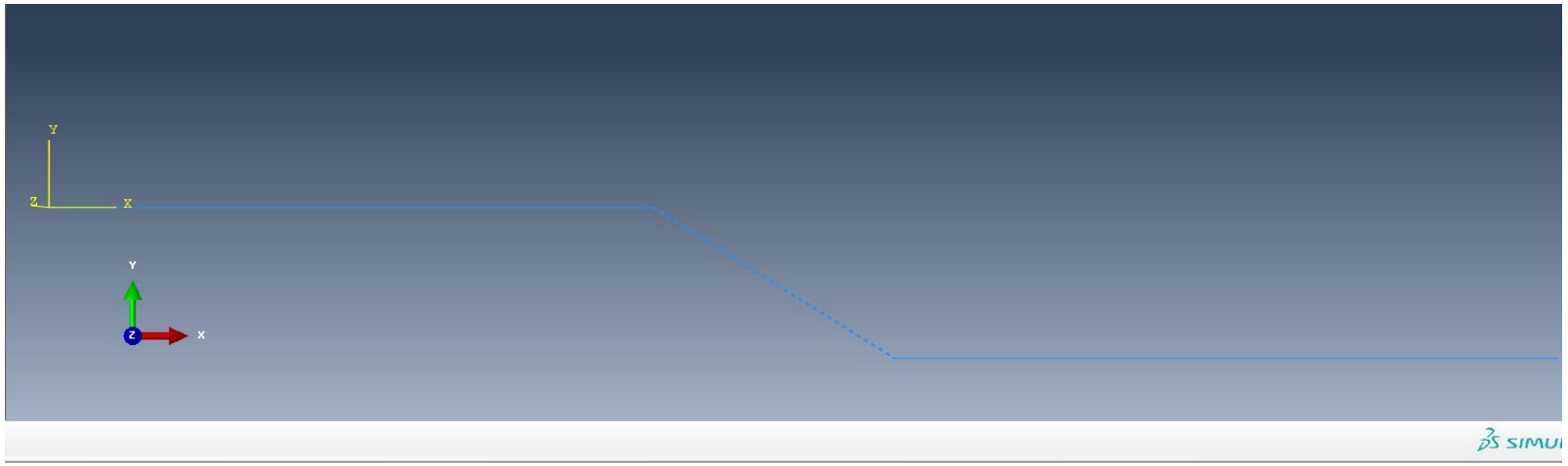
ENTER

Далее соединим наши балки шарниром. Для этого вначале в модуле INTERACTION выберем инструмент Create Wire Feature:

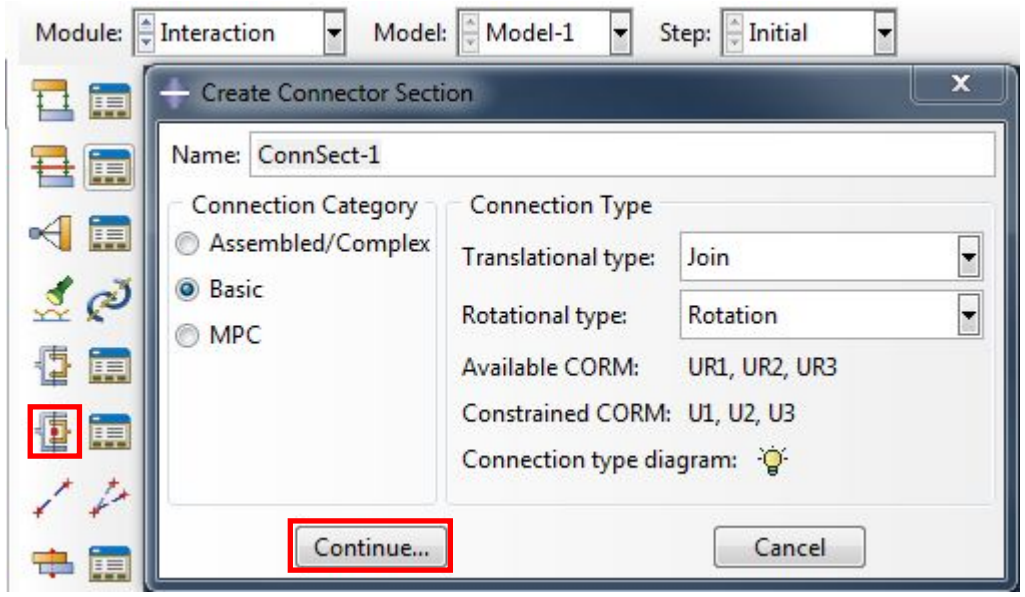


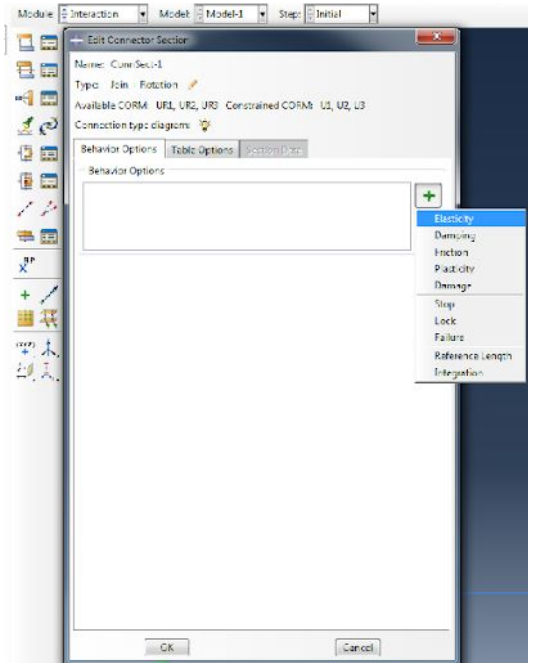


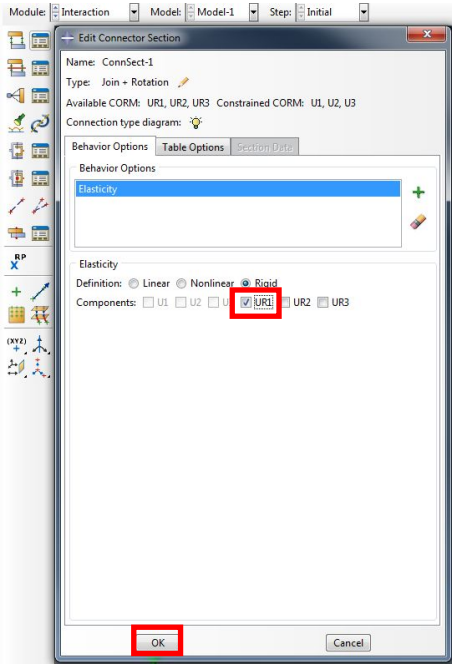




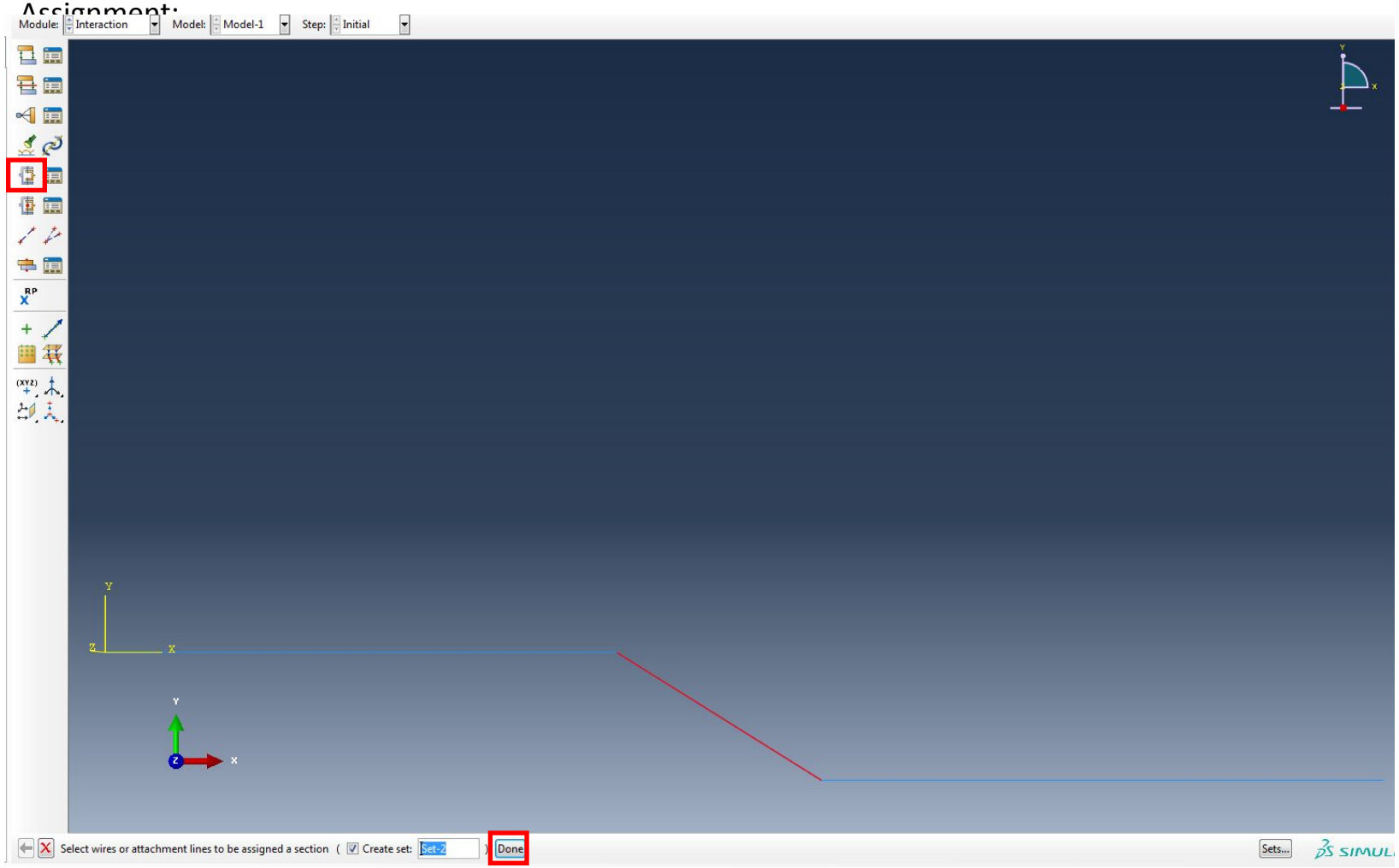
Далее создадим сечение для коннектора. Для этого в модуле INTERACTION выберем инструмент Create Connector Section и укажем следующие параметры:

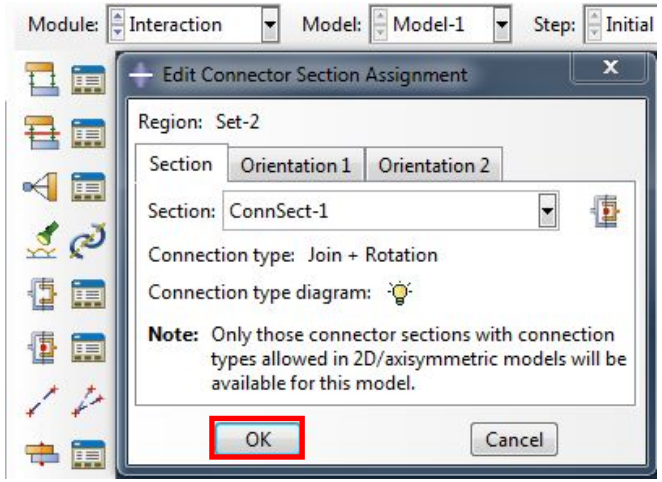


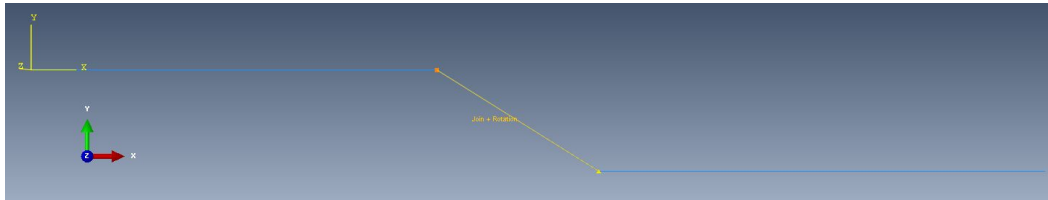




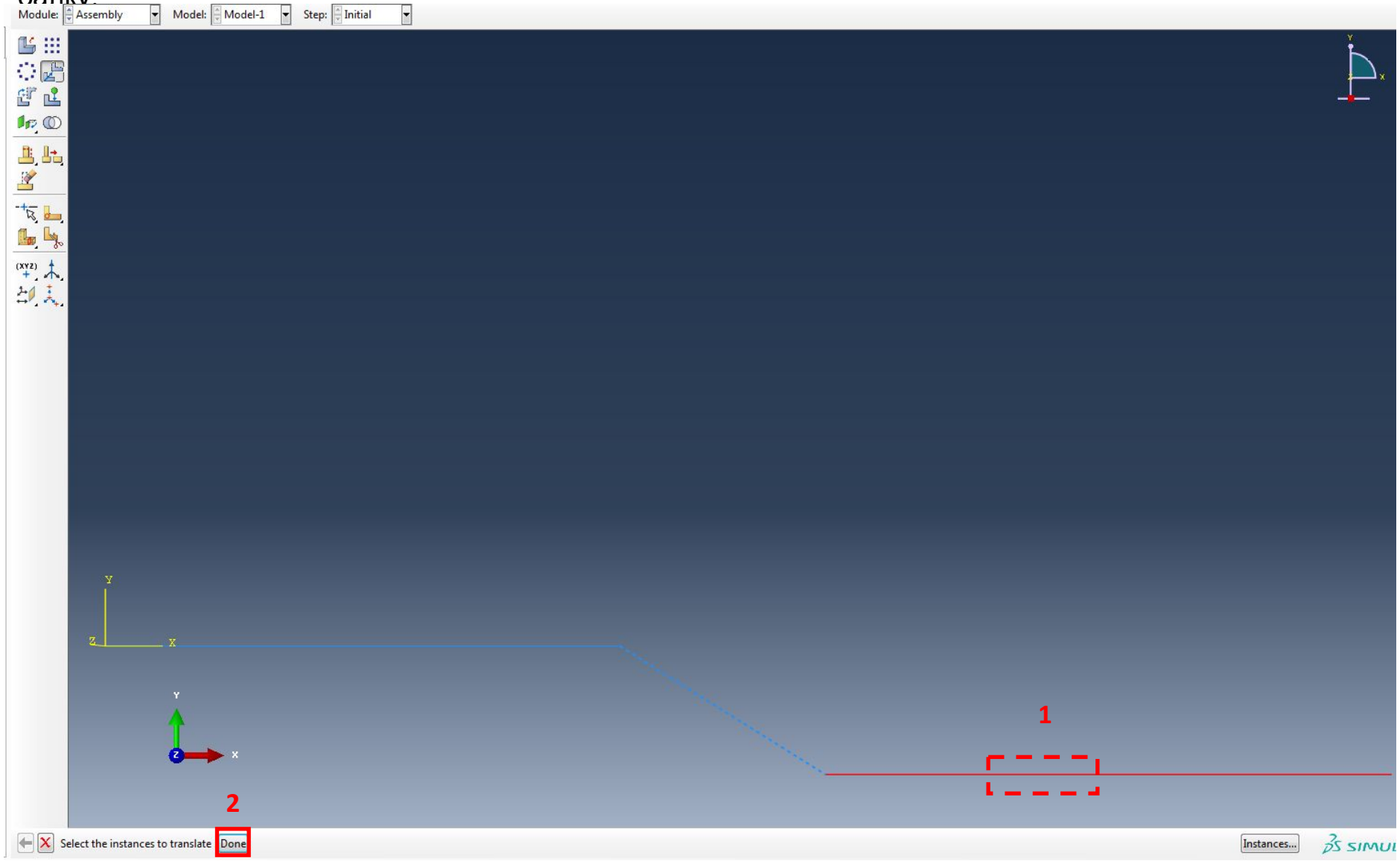
Теперь назначим это сечение. Для этого в модуле INTERACTION выберем инструмент Create Connector Assignment:



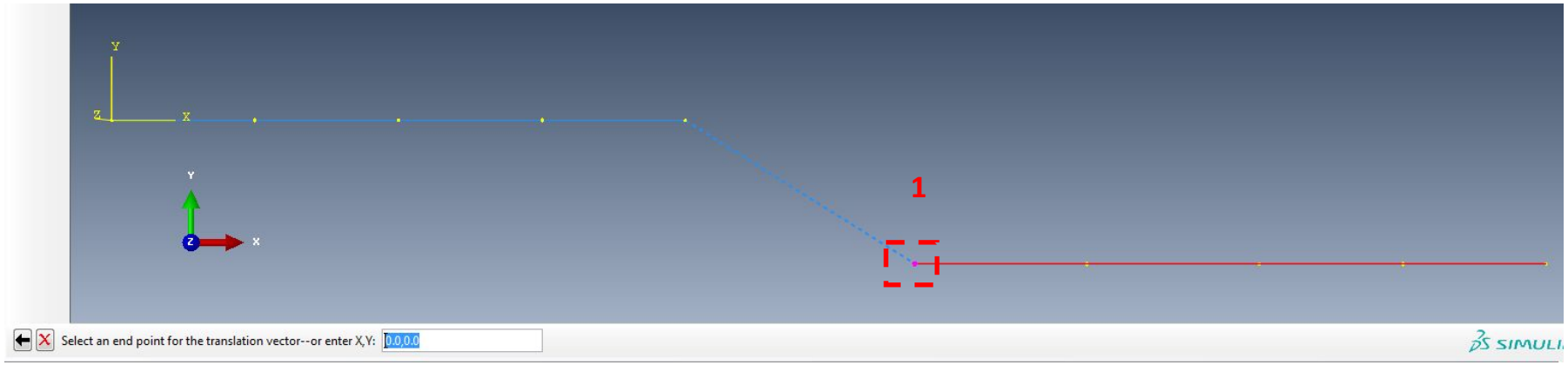




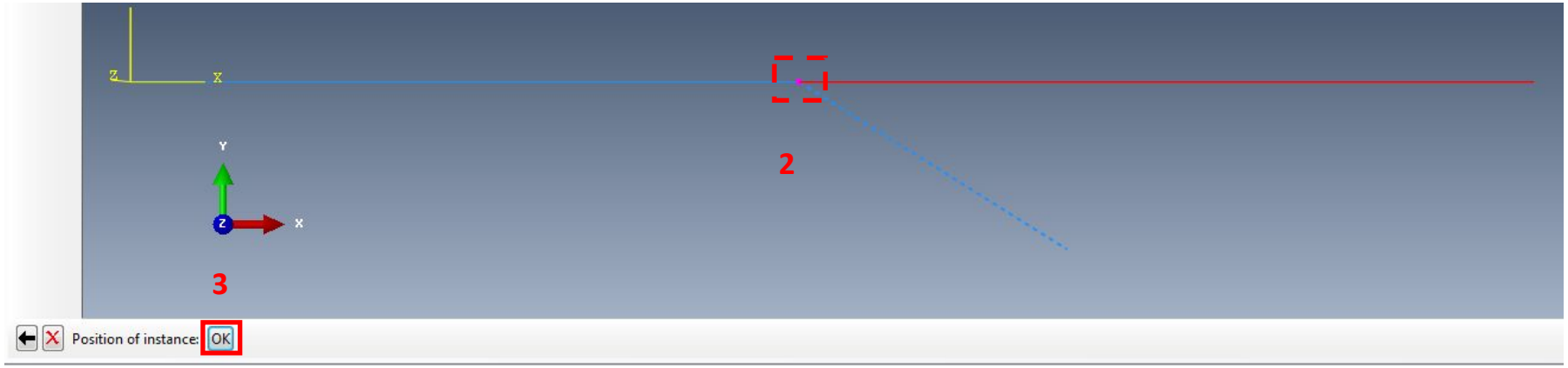
Далее совместим наши балки. Для этого в модуле ASSEMBLY выберем инструмент Translate Instance и выберем правую балку:



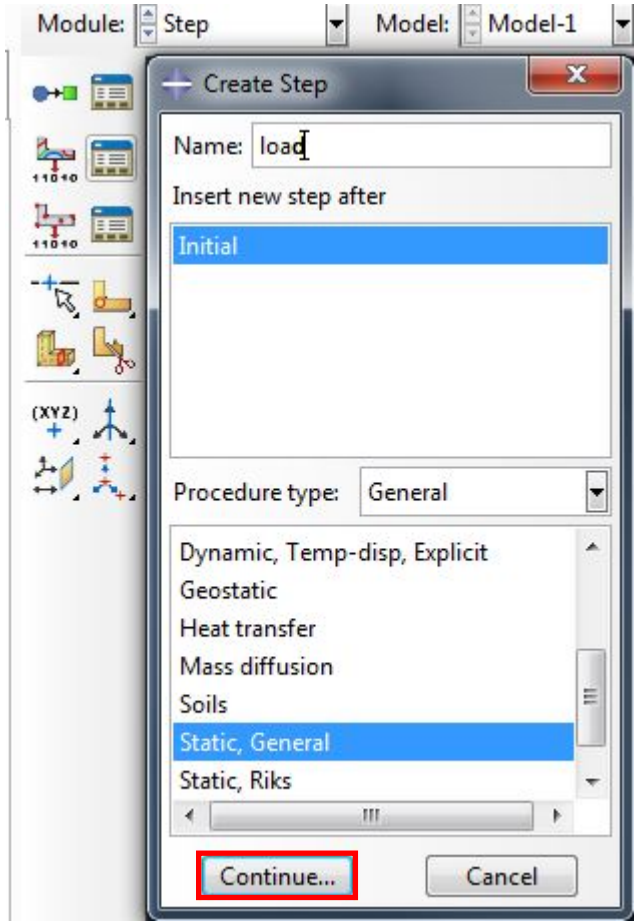
Выберем в качестве начальной точки левый узел нижней нашей балки:

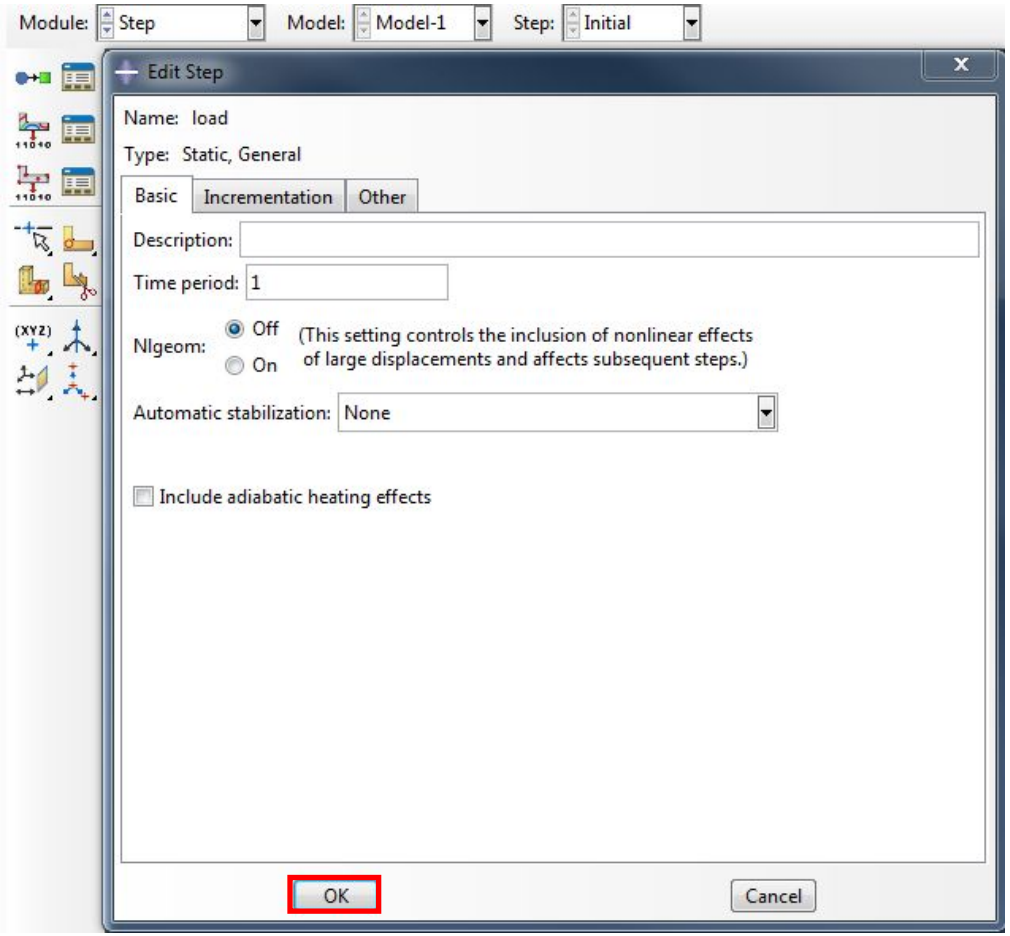


В качестве конечной точки выберем правый узел верхней балки:



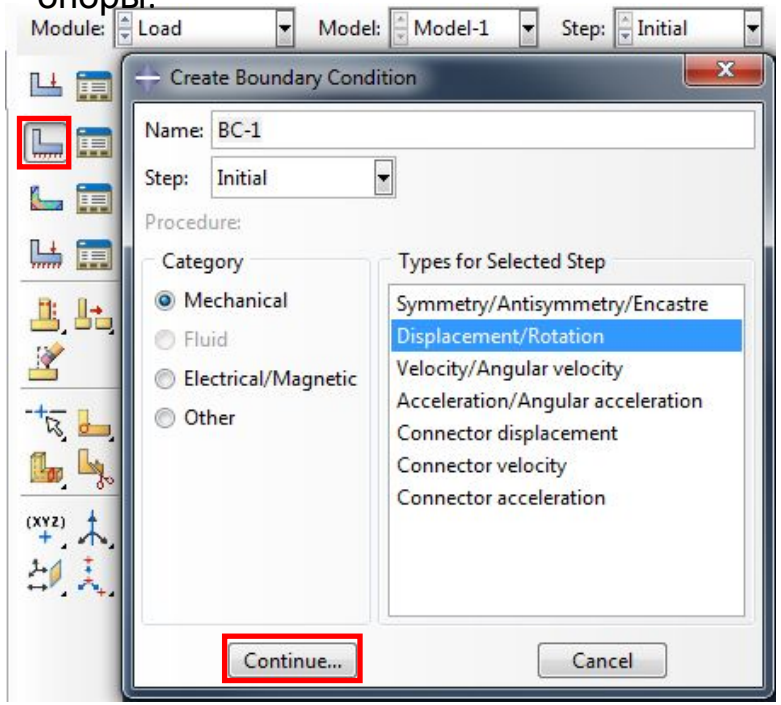
Шаг 11. Создадим шаг нагружения для нашей задачи. Для этого в модуле Step выберем инструмент Create Step:



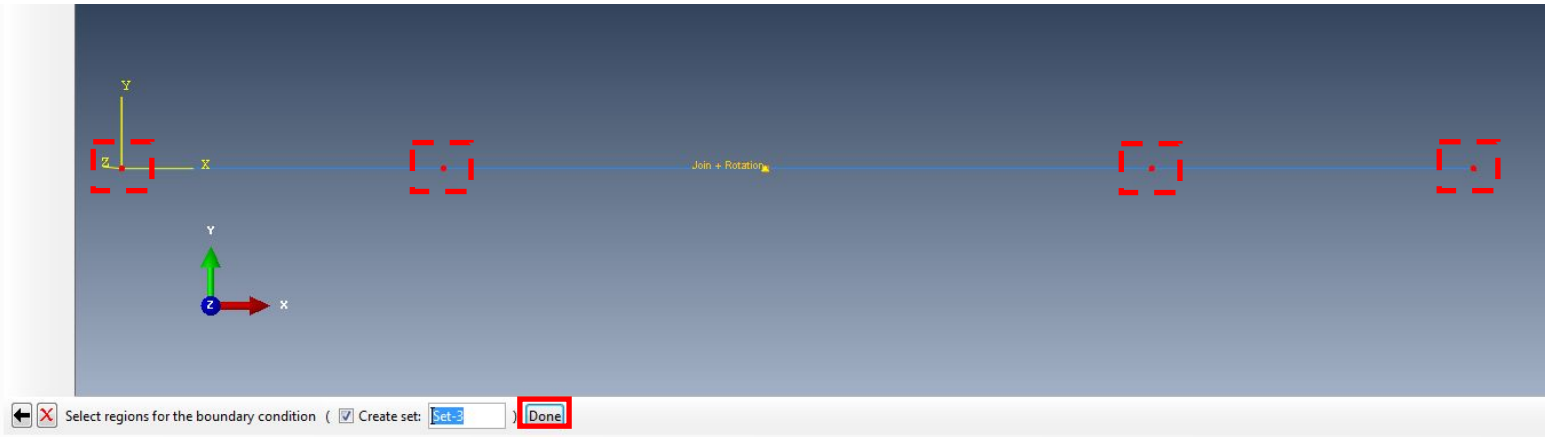


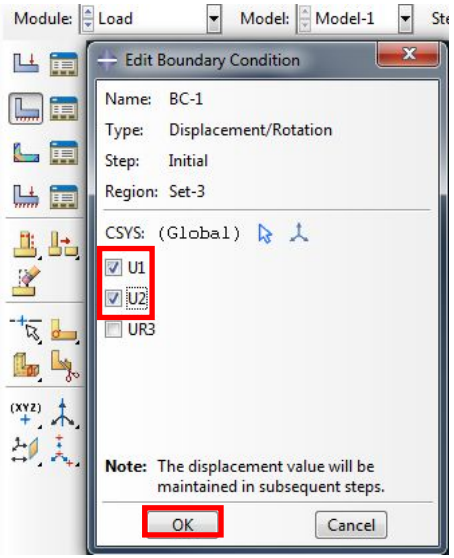
Шаг 12. Добавим граничные условия в нашу модель. Для этого в модуле Load выберем инструмент Create Boundary Condition:

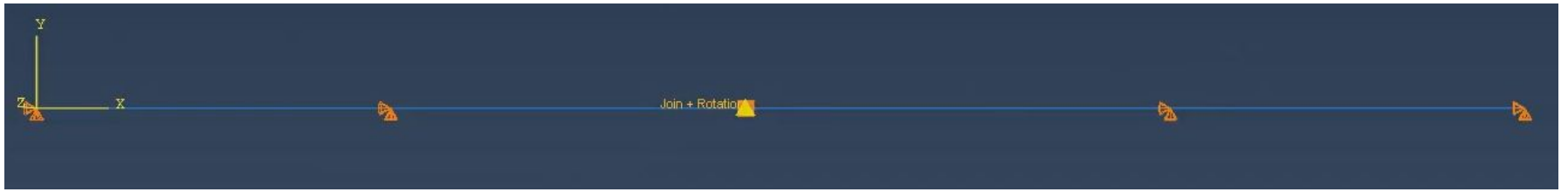
1) Зададим опоры:



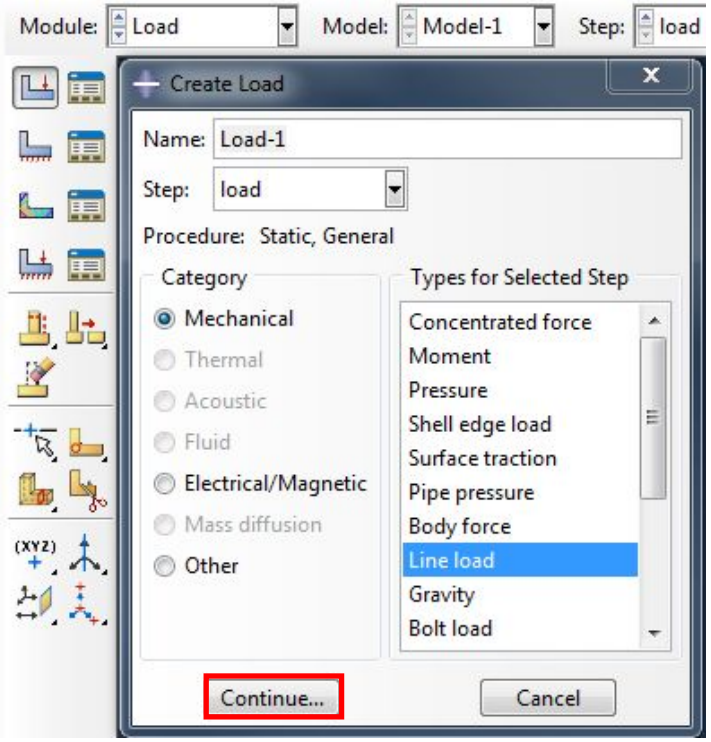
Далее поочередно выбираем четыре узла и нажимаем Done:

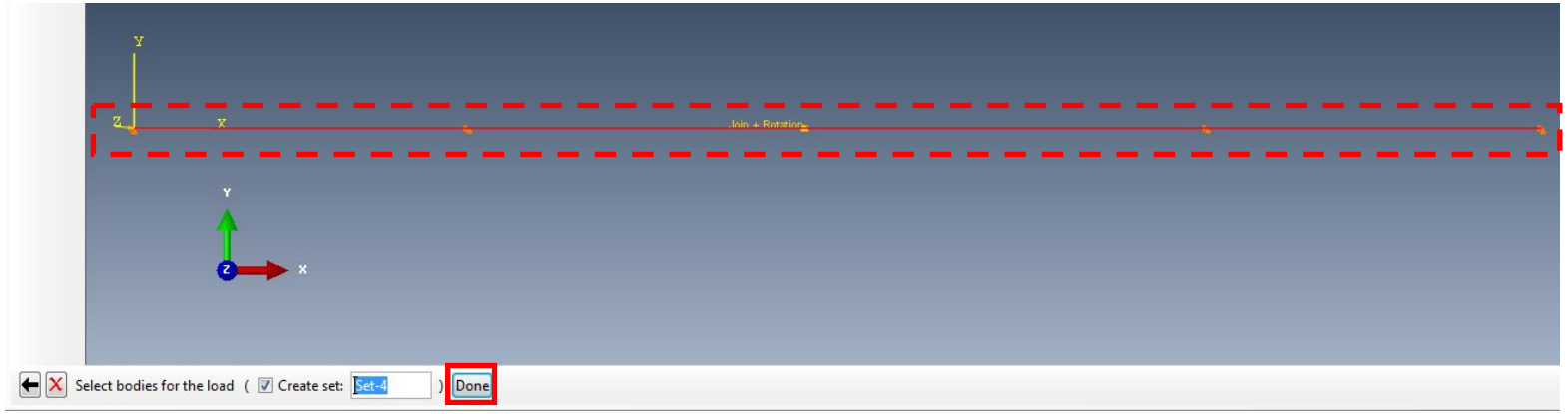


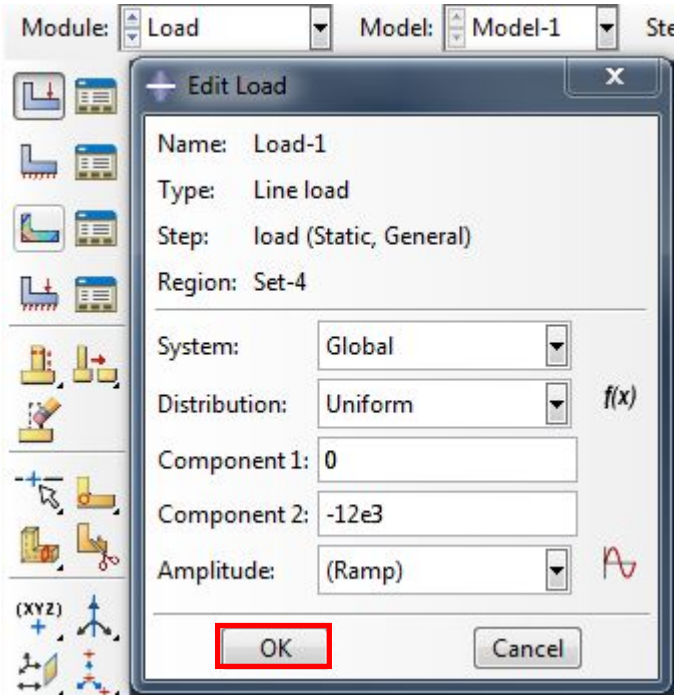


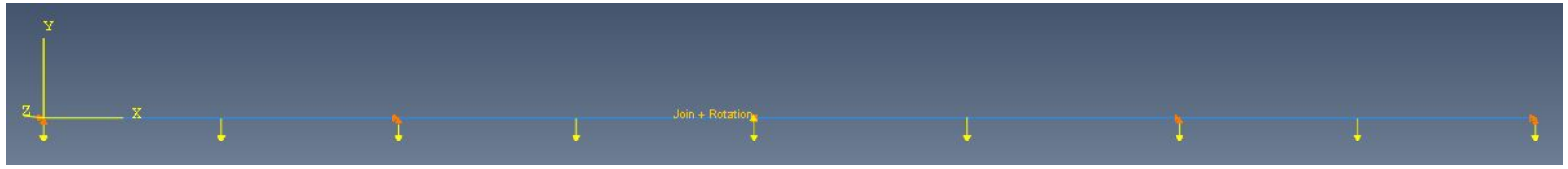


2) Зададим нагрузку на многопролетную балку:

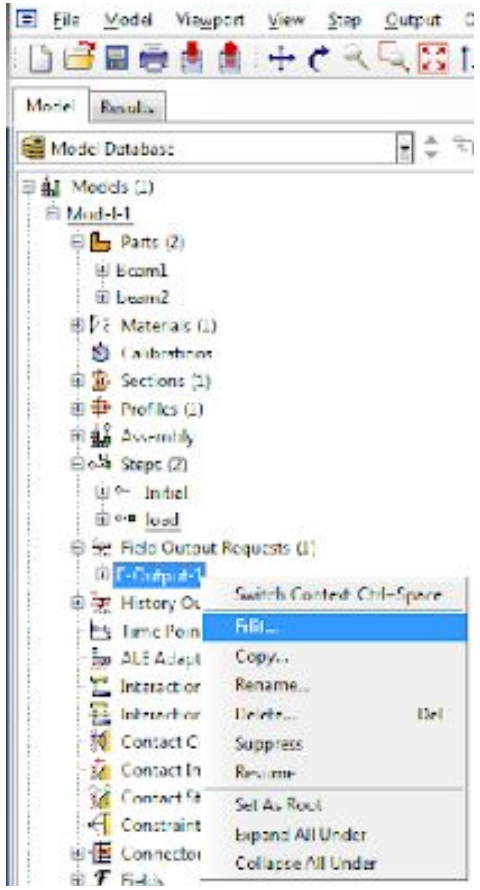


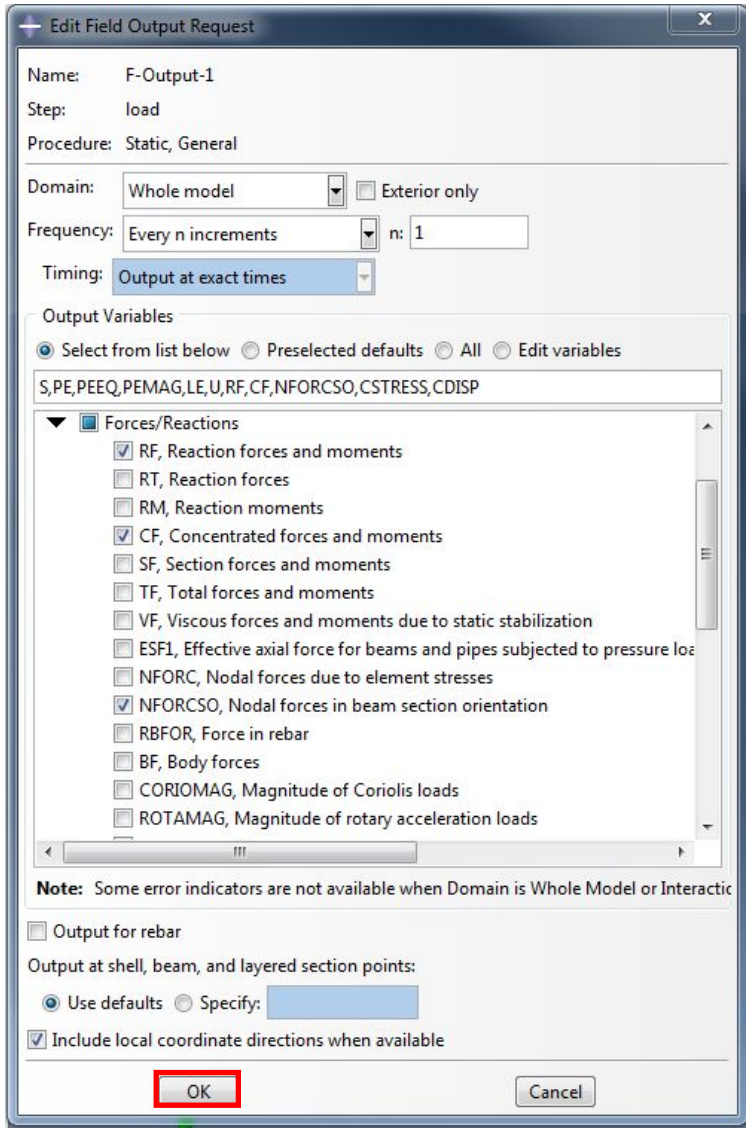



Module: Load
Model: Model-1
Edit Load
Name: Load-1
Type: Line load
Step: load (Static, General)
Region: Set-4
System: Global
Distribution: Uniform
f(x)
Component 1: 0
Component 2: -12e3
Amplitude: (Ramp)
OK
Cancel

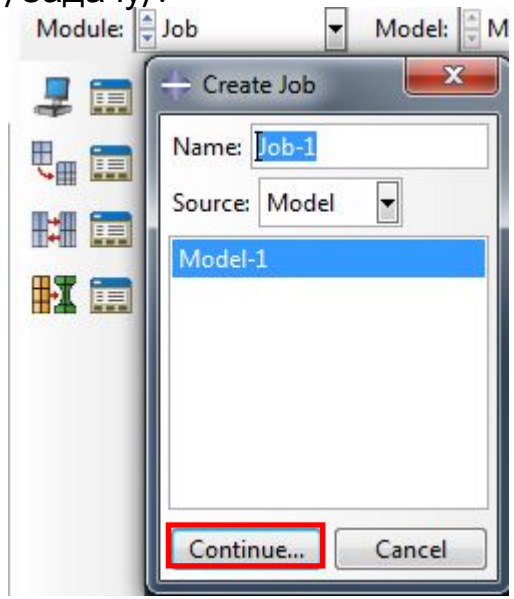

Y
Z
X
Join + Rotation

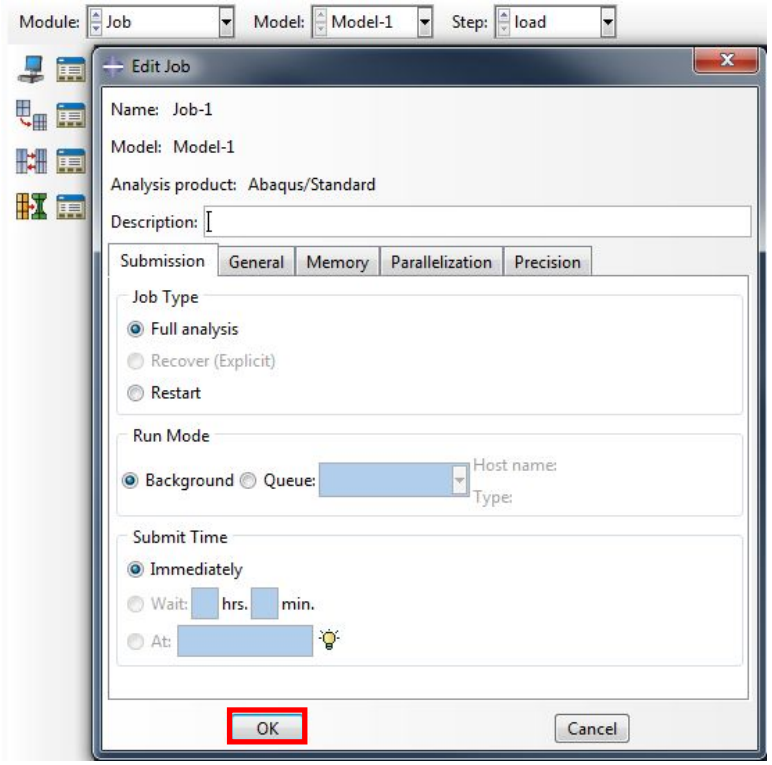
Шаг 13. Добавим дополнительную переменную для вывода, открыв ветвь Field Output Requests>F-Output-1 и нажав ПКМ по F-Output-1 -> Edit:

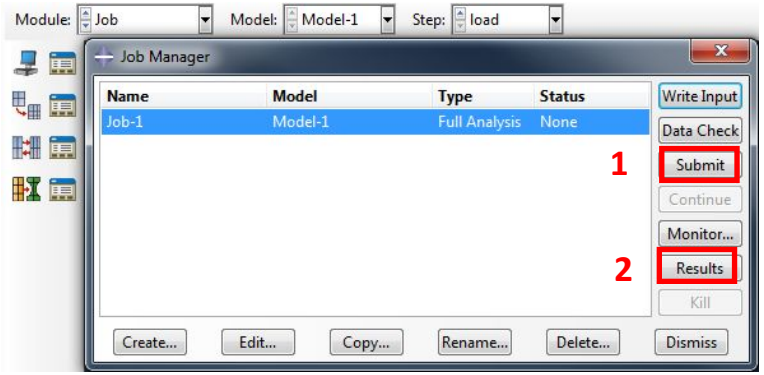




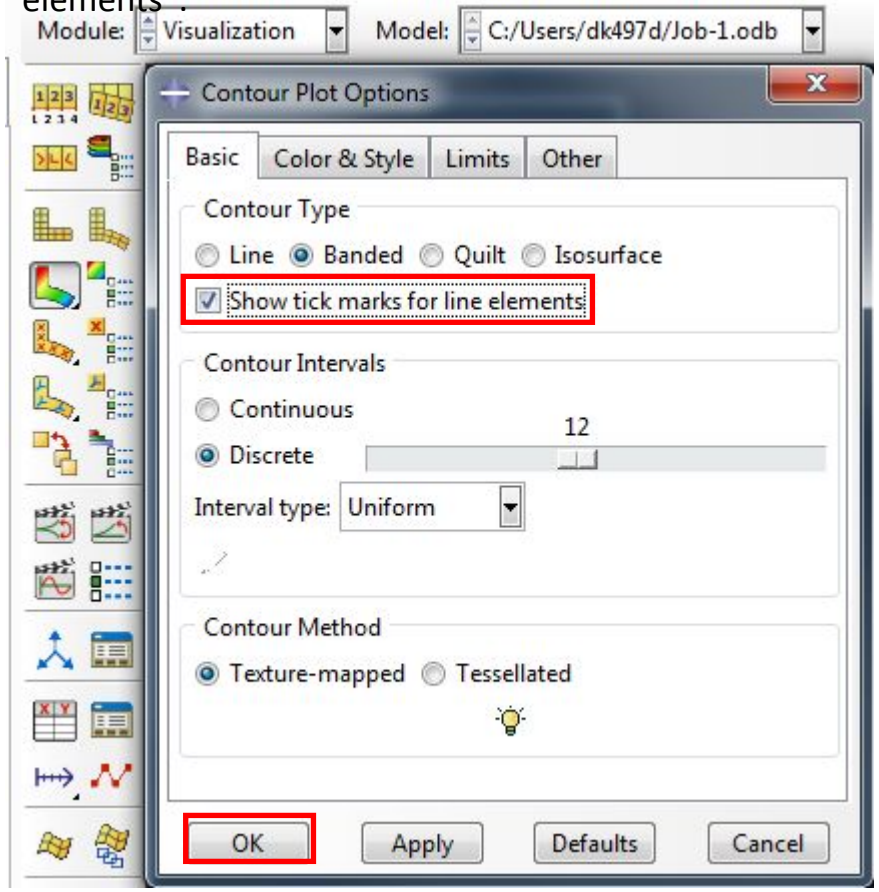
Шаг 14. Создадим Job /Задачу/:



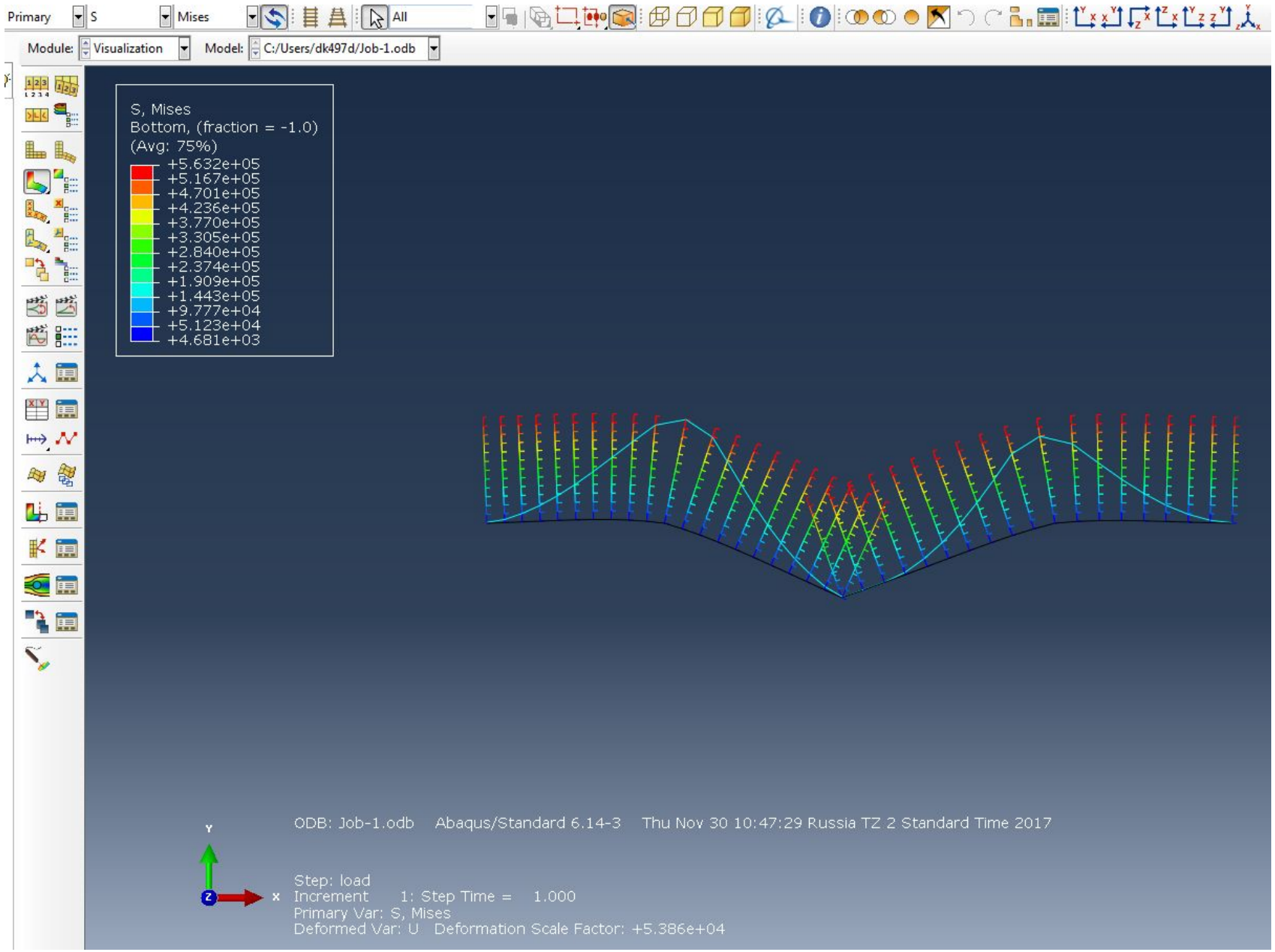




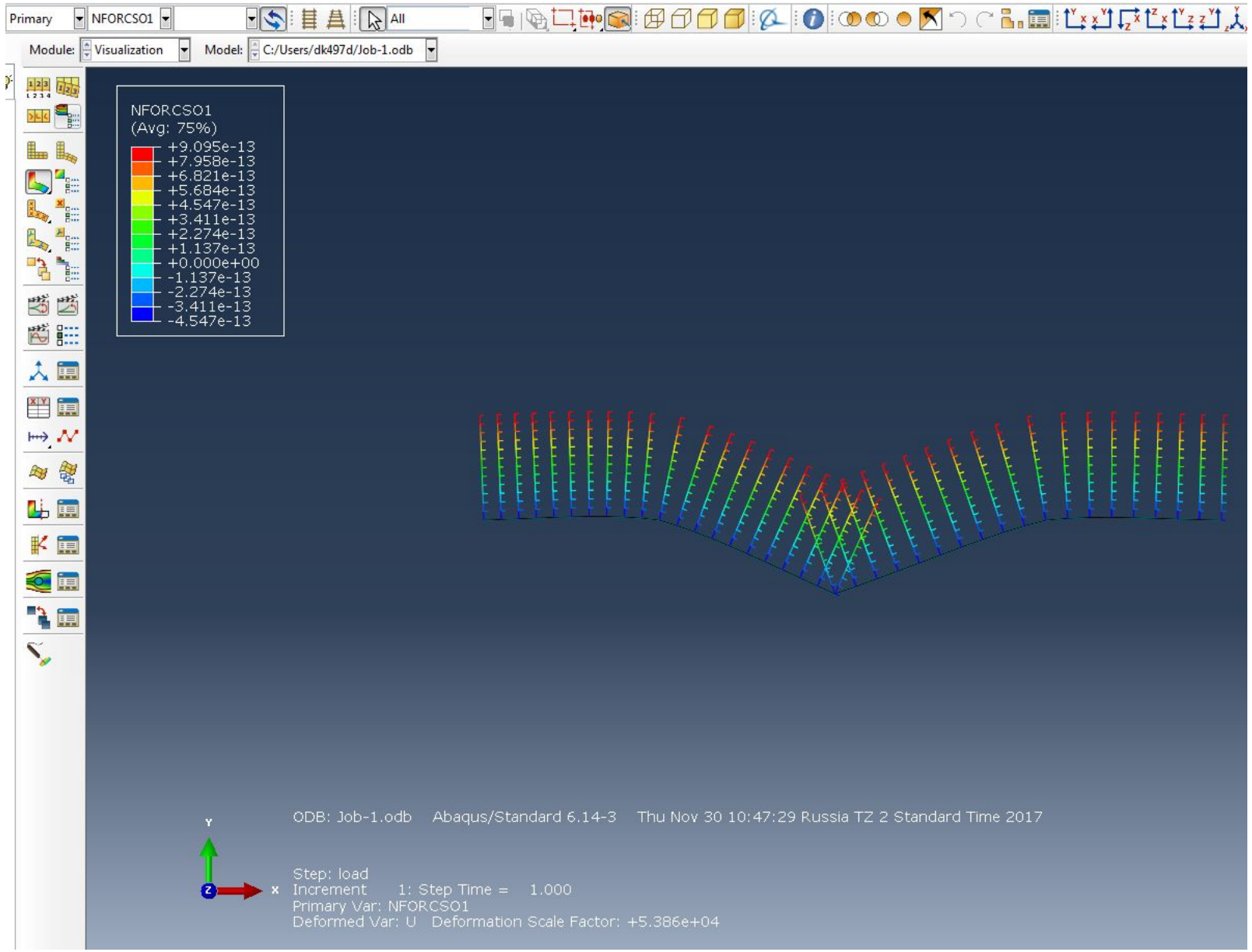
Шаг 15. В модуле Visualization включим опцию “Show tick marks for line elements”:



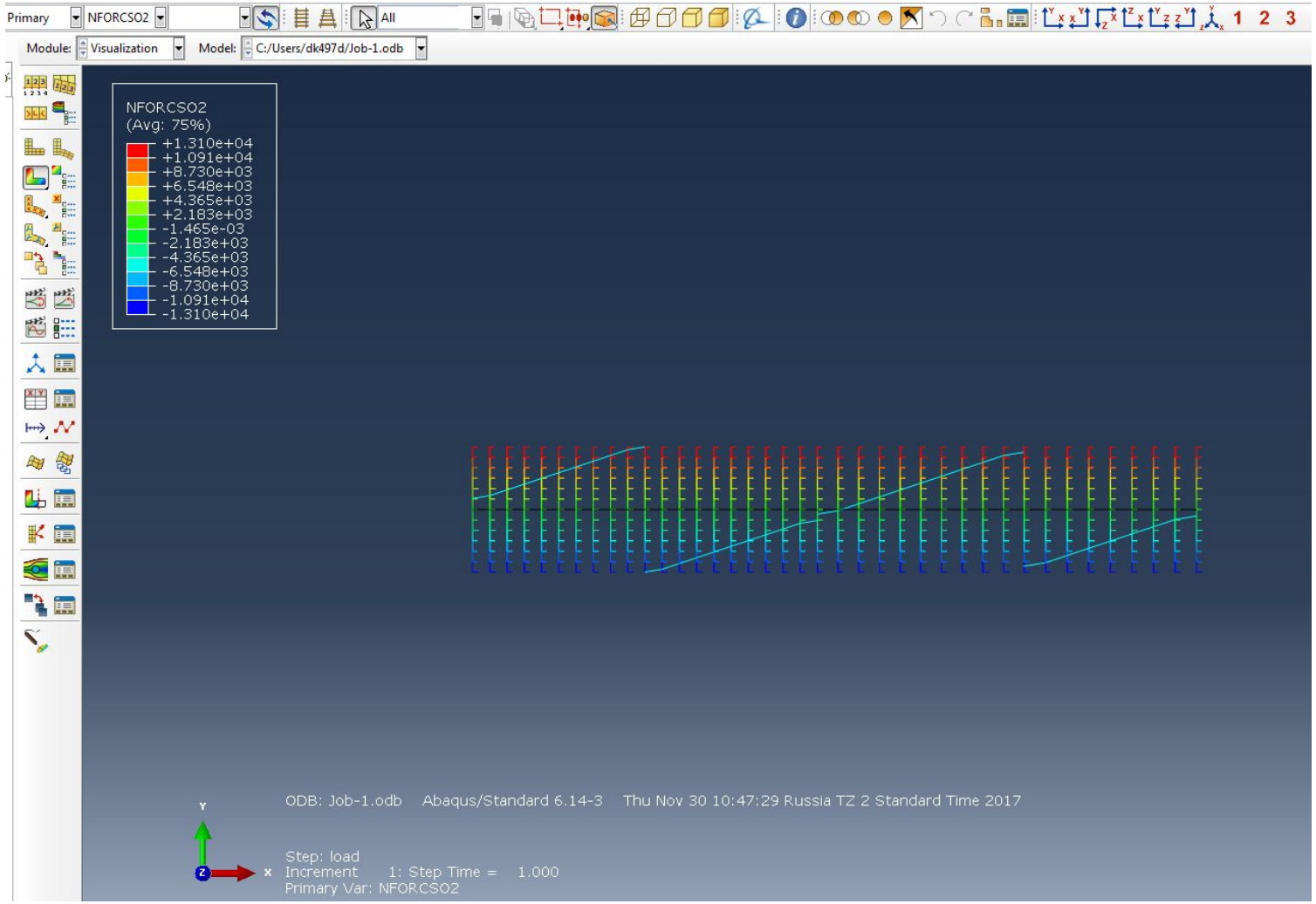
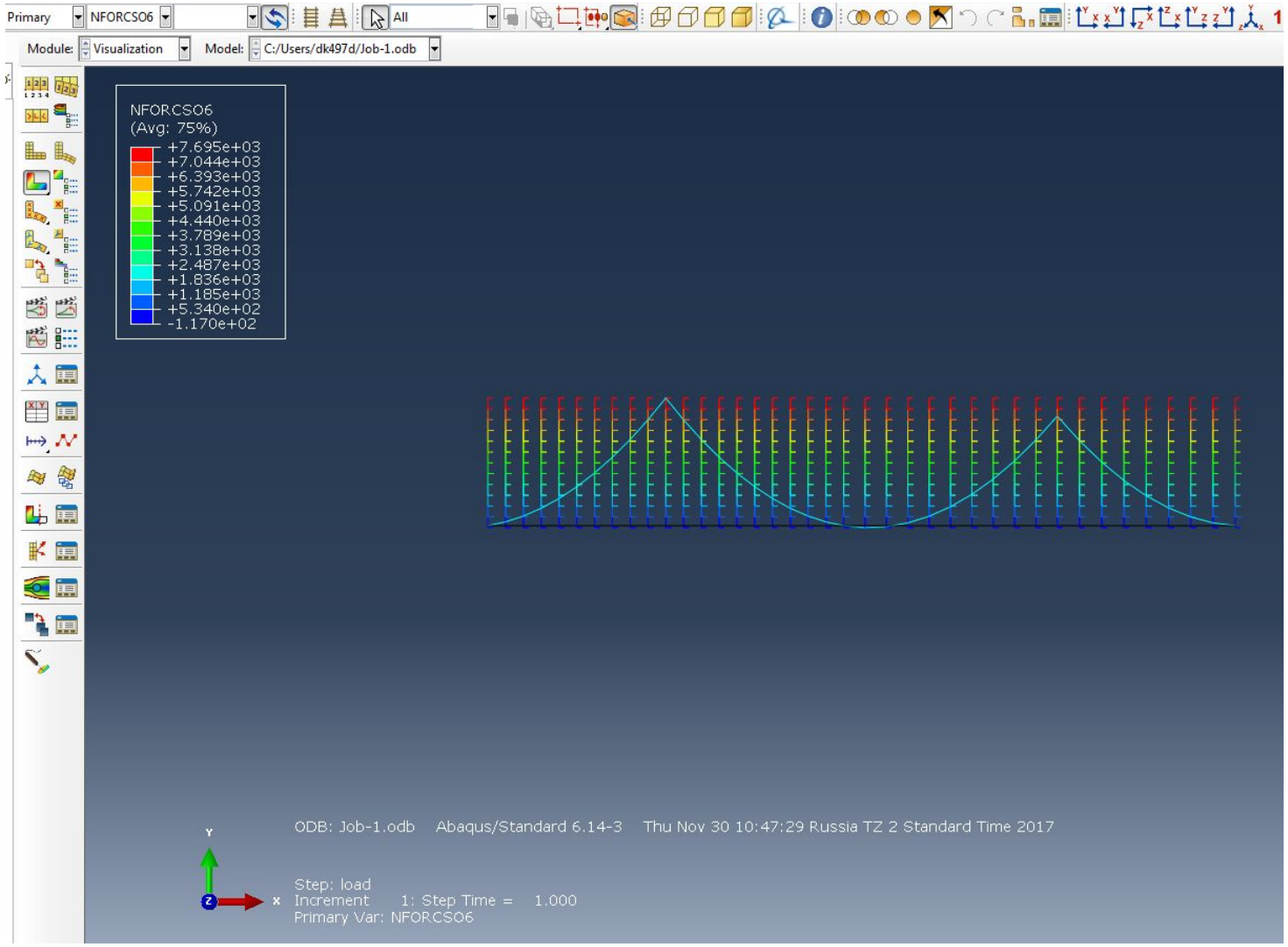
Выведем напряжения по Мизесу:



- Axial forces /Осевые силы/ в балке:



- Shear forces /Перерезывающие силы/ в балке:



- Moments /Моменты сил/ в балке: