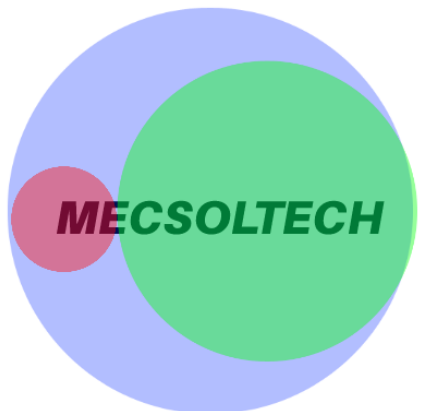
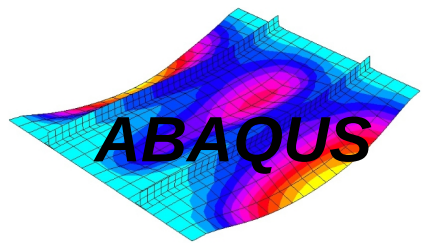


Análise de vigas

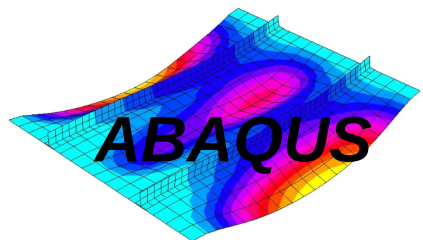
Elemento 1D





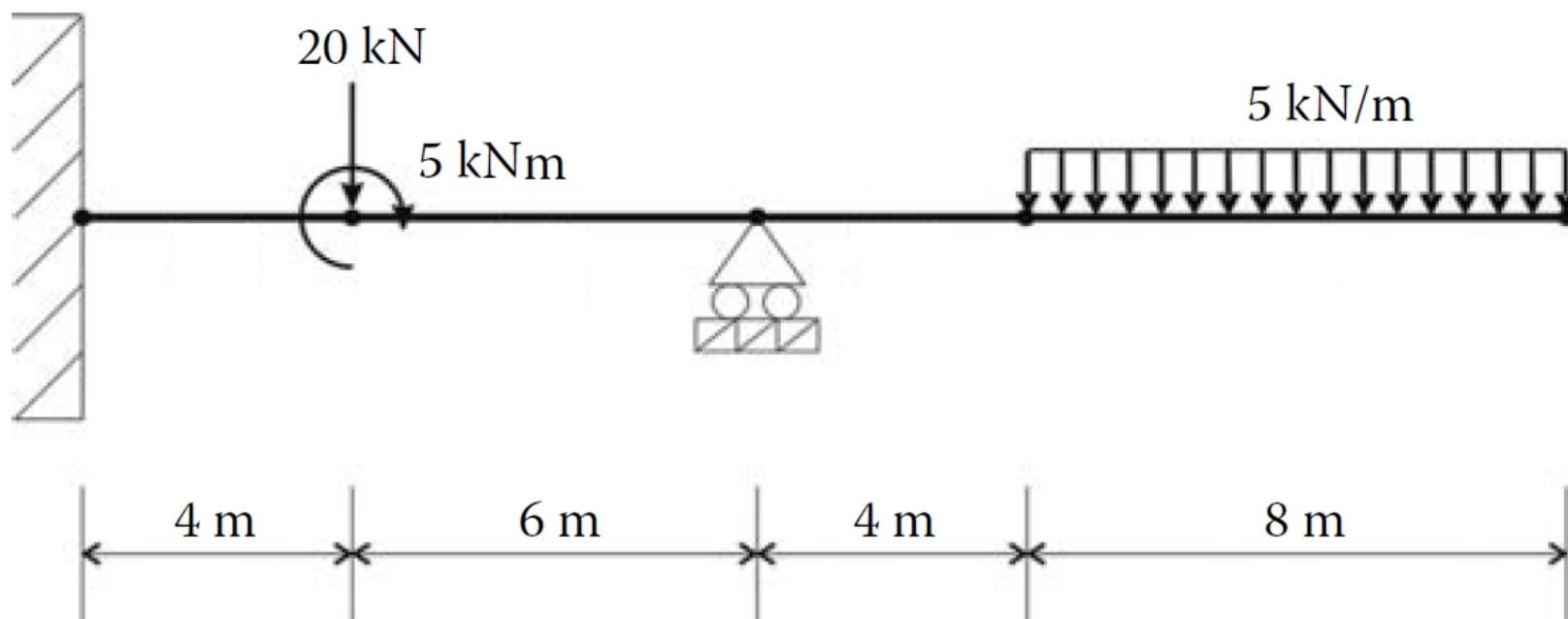
Objetivo

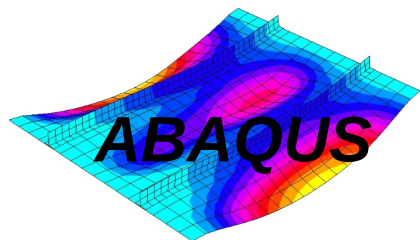
Realizar análise de elementos finitos em um modelo de viga discretizado em elementos 1D.



Estudo de caso

Faça a análise estrutural em ABAQUS da viga ilustrada na figura. Utilize elementos de viga (1D) para a discretização. Considere a seção transversal da viga maciça e retangular, com altura $H = 263,2148 \text{ mm}$ e base $B = 131,6074 \text{ mm}$. O material é aço com módulo de elasticidade $E = 200 \text{ GPa}$ e coeficiente de Poisson $\nu = 0,3$.

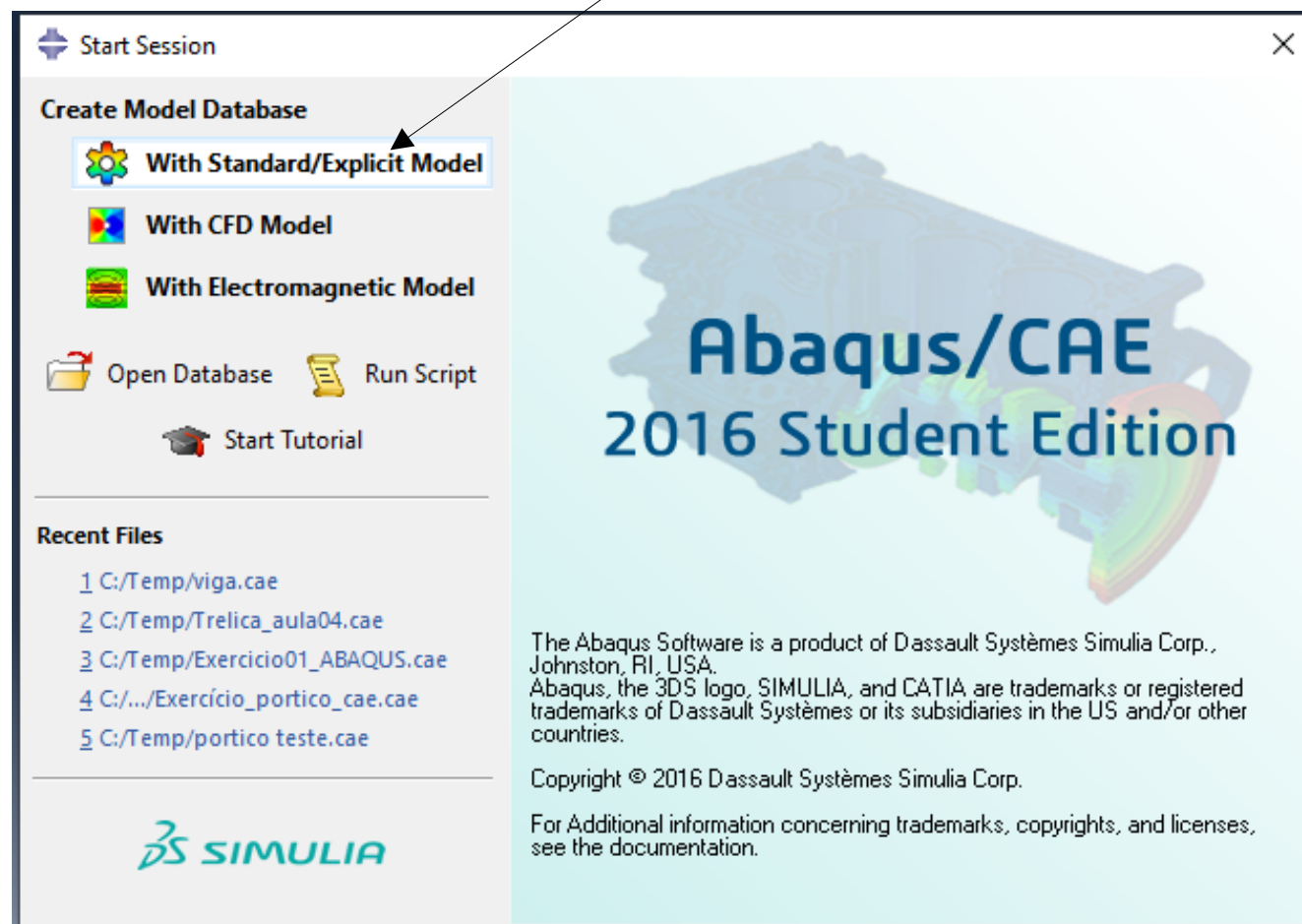


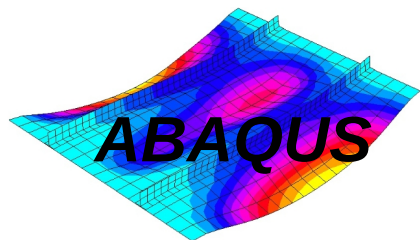


Passo 1

Inicie o ABAQUS

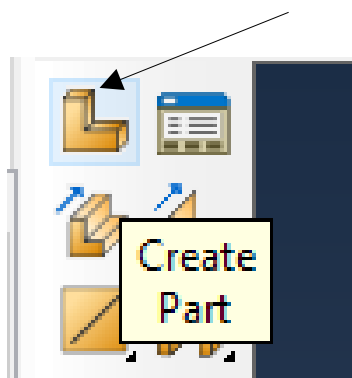
Em seguida
clique aqui





Passo 2

Primeiro
Clique aqui



Em seguida, na janela **Create Part** selecione as definições conforme a imagem ao lado.

Create Part

Name:

Modeling Space

☐ 3D ☒ 2D Planar ☐ Axisymmetric

Type

☒ Deformable
☐ Discrete rigid
☐ Analytical rigid
☐ Eulerian

Options

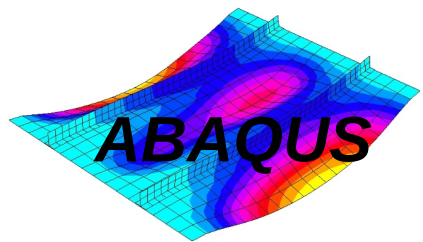
None available

Base Feature

☐ Shell
☒ Wire
☐ Point

Approximate size:

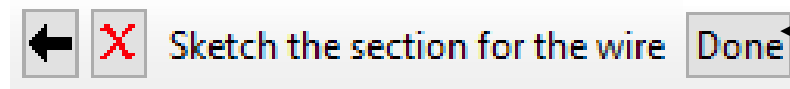
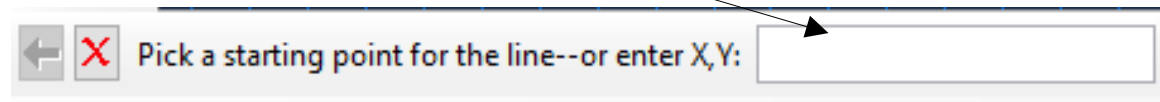
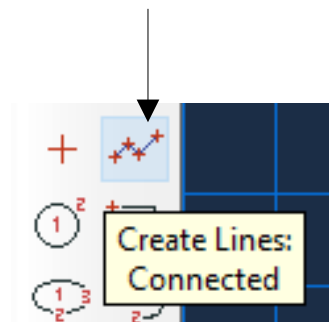
Continue... Cancel



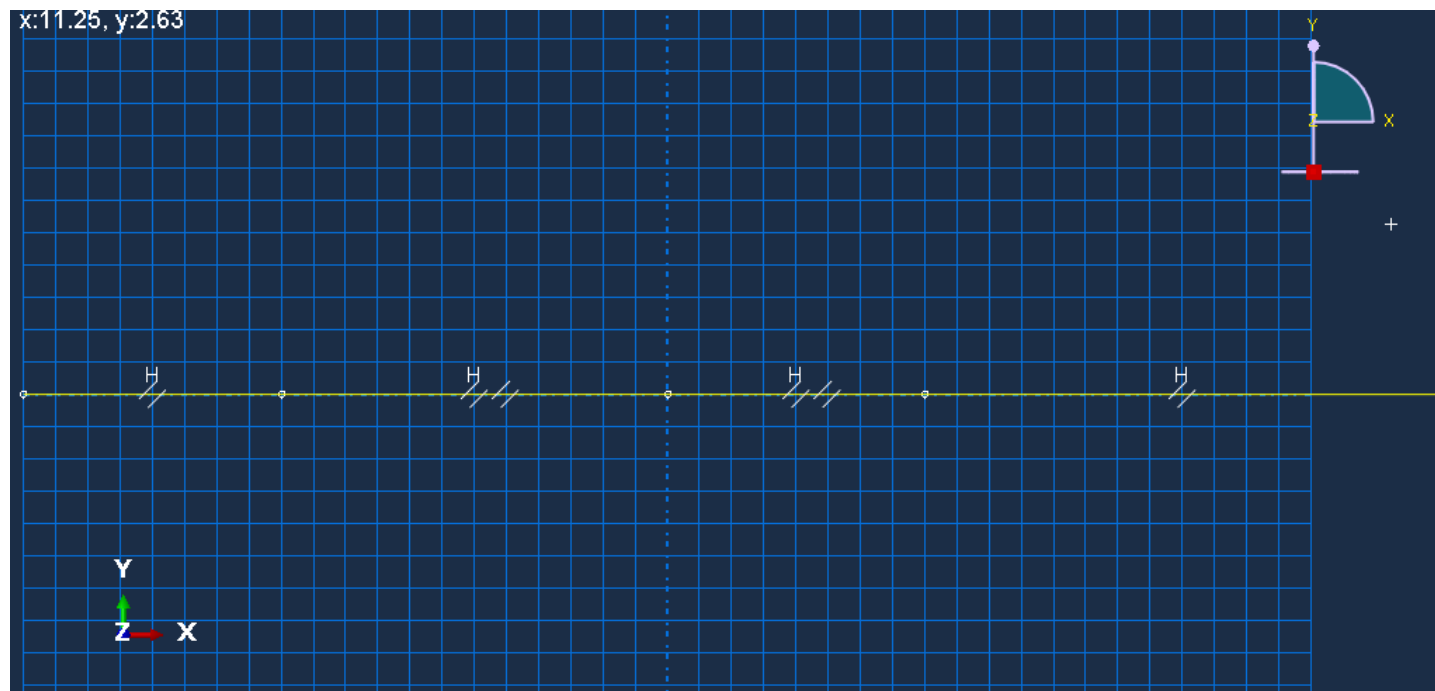
Passo 3

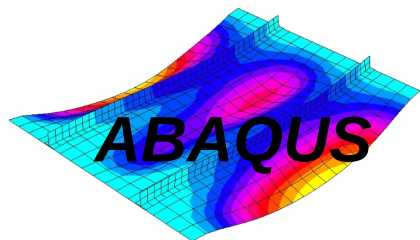
Em seguida, na barra de coordenadas faça:
-10,0 Enter;-6,0 Enter; 0,0 Enter; 4,0 Enter; 12,0 Enter e Esc.

No sketcher
menu clique aqui



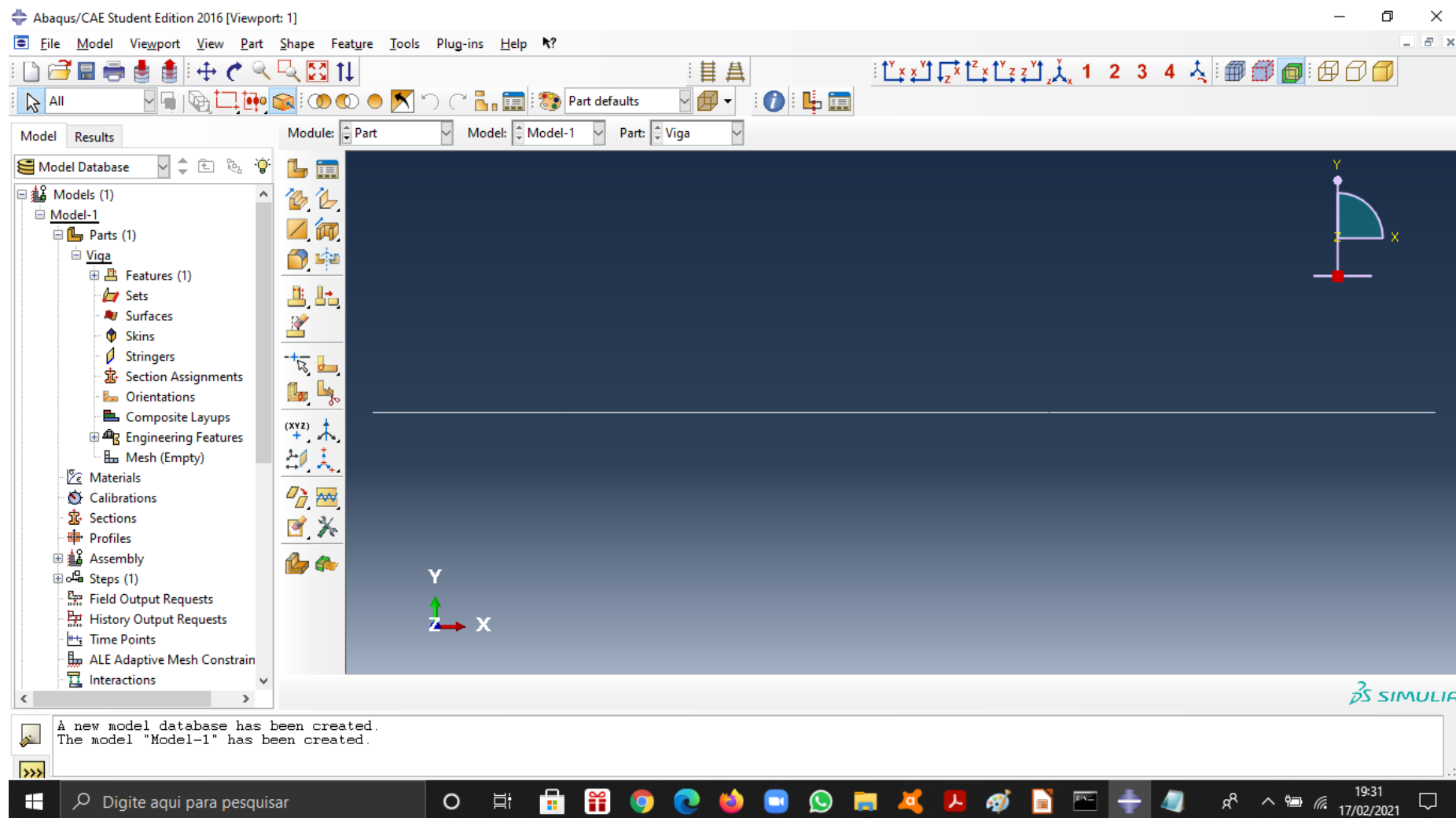
Finalmente,
clique em Done.



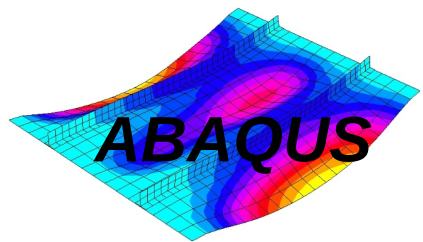


Passo 4

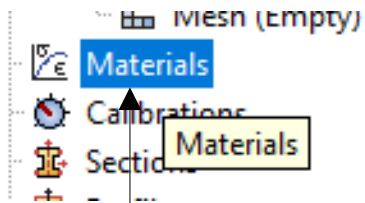
Definir material



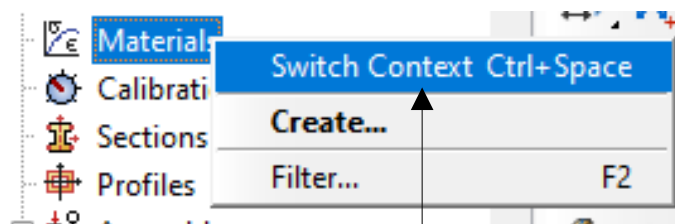
Prof. Me. Hélio Guerrini Filho



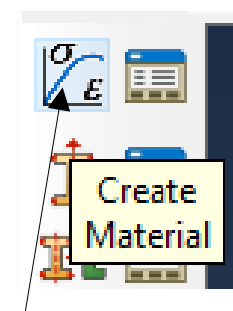
Passo 4



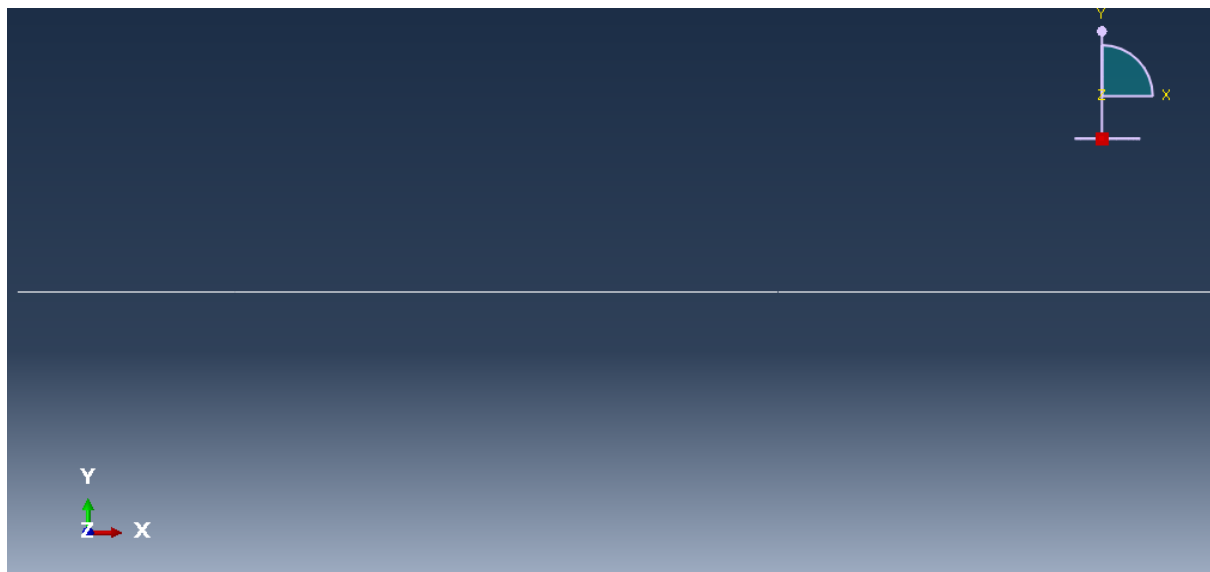
Clique com o botão direito em **Materials** na árvore do modelo.



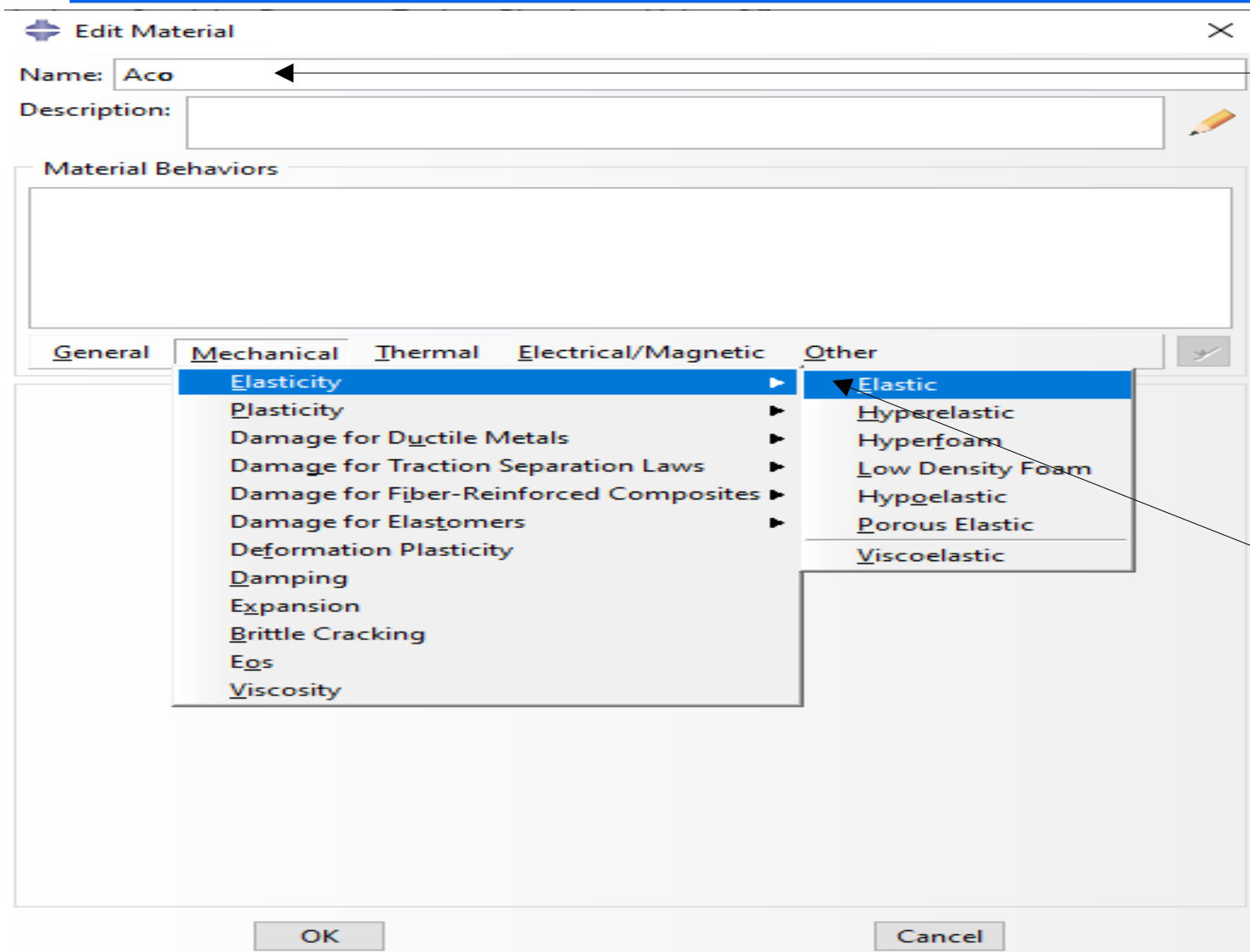
Em seguida, clique aqui.



Na barra de comandos habilitada, clique em **Create Material**.



Passo 5



Edit Material

Name:

Description:

Material Behaviors

General Mechanical Thermal Electrical/Magnetic Other

Elasticity ▶ Elastic
Plasticity ▶ Hyperelastic
 Damage for Ductile Metals ▶ Hyperfoam
 Damage for Traction Separation Laws ▶ Low Density Foam
 Damage for Fiber-Reinforced Composites ▶ Hypogelastic
 Damage for Elastomers ▶ Porous Elastic
 Deformation Plasticity Viscoelastic
Damping
Expansion
Brittle Cracking
Eos
Viscosity

OK Cancel

Nome do material.

Clique aqui para o material elástico

Edit Material

Name: Aco

Description:

Material Behaviors

Elastic

General Mechanical Thermal Electrical/Magnetic Other

Elastic

Type: Isotropic

☐ 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	200e9	0.3

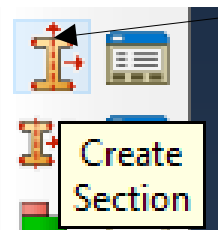
OK Cancel

Preencha as propriedades do material (aço).

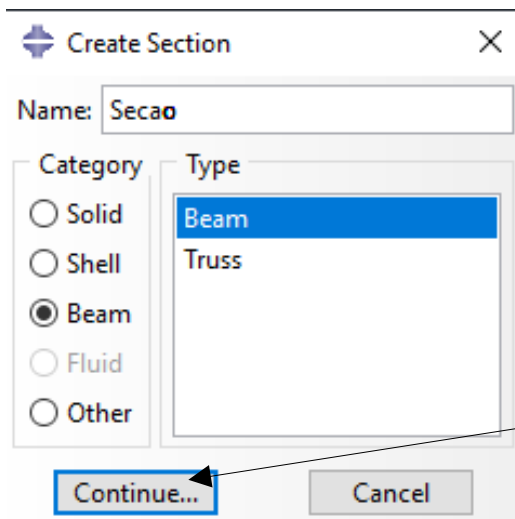
Em seguida, clique aqui.

Passo 6

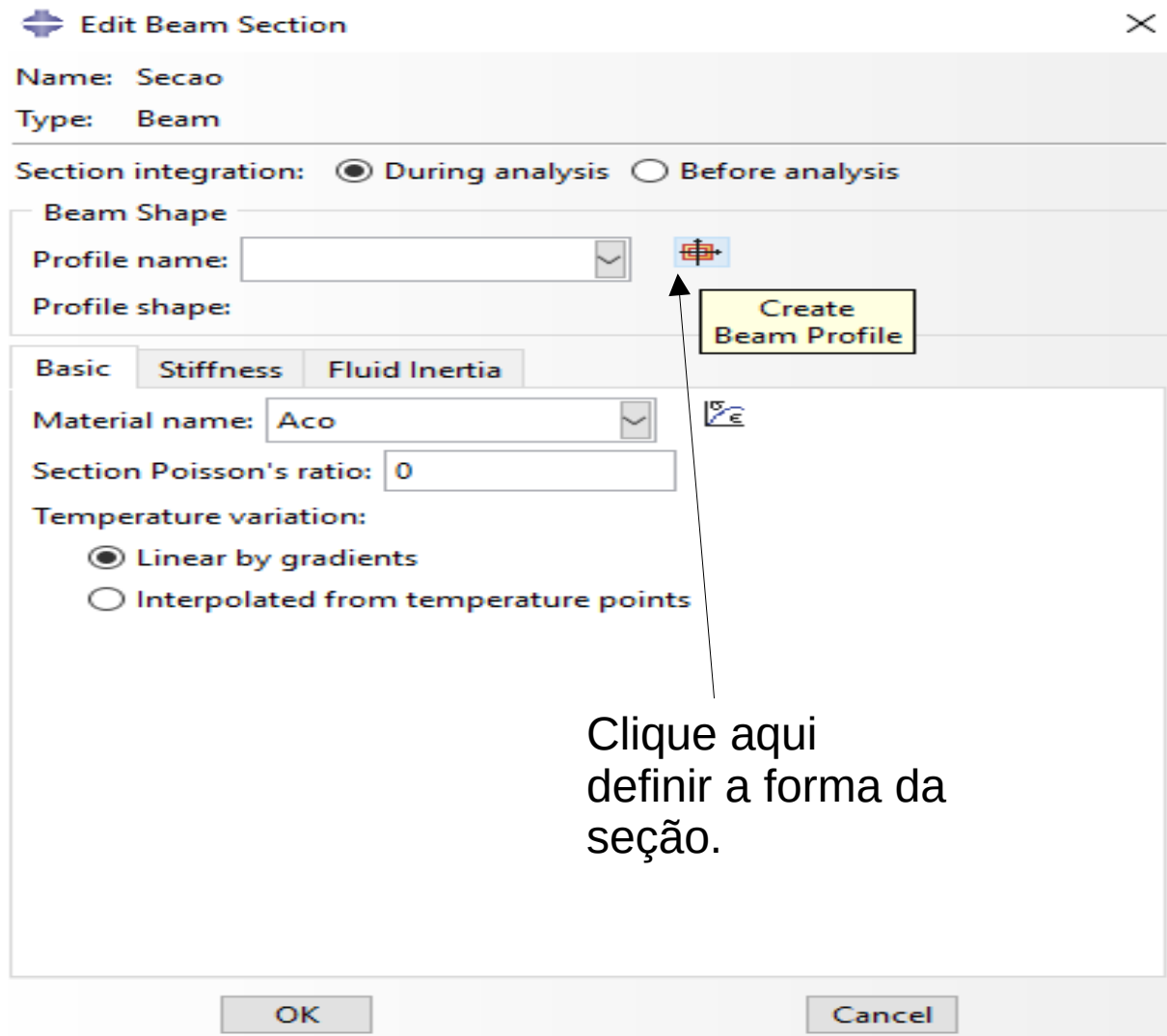
Clique aqui para criar a seção.



Na janela **Create Section** selecione as definições conforme ilustra a figura.



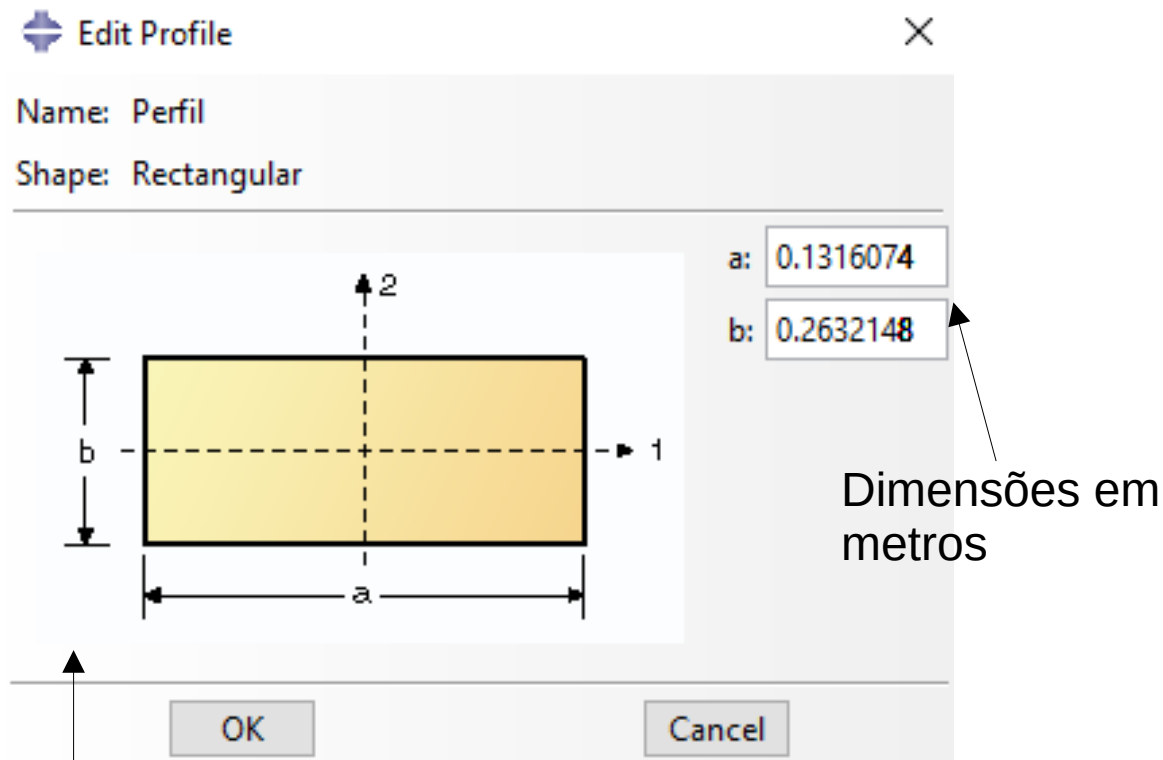
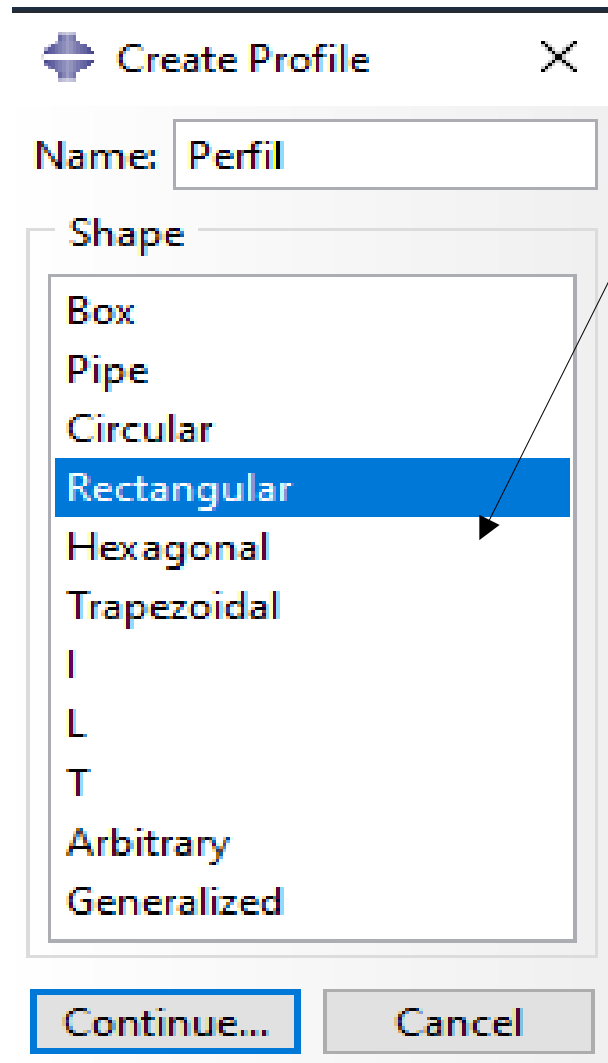
Em seguida, clique em **Continue...**



Clique aqui definir a forma da seção.

Passo 7

Defina as informações conforme a figura ao lado.

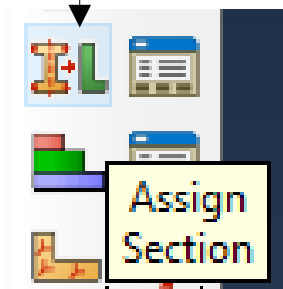


Defina as informações conforme a figura acima. Em seguida, clique Ok. Finalmente, em **Edit beam profile** e clique Ok.

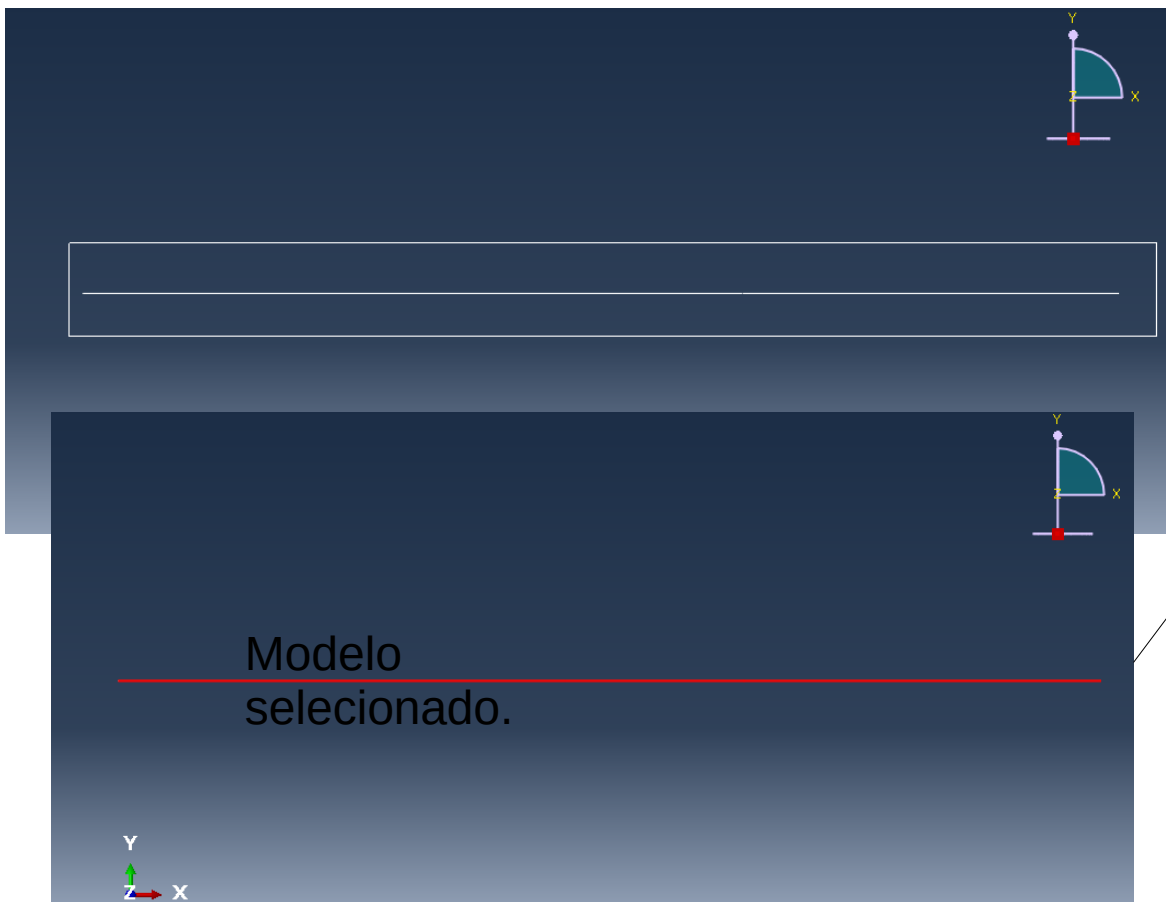
Passo 8

Selecionar o modelo para atribuir a seção.

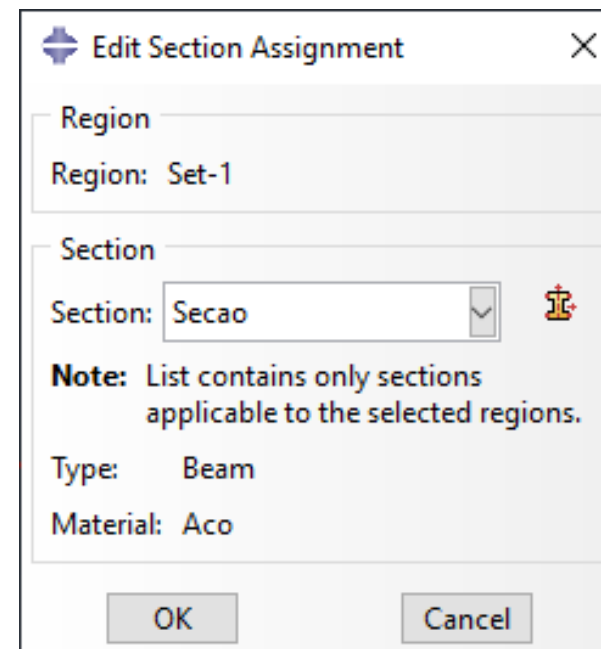
Clique aqui para atribuir a seção para o modelo de viga.

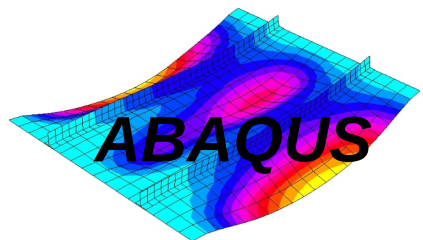


← ✖ Select the regions to be assigned a section (☒ Create set: Set-1) Done



Em seguida, clique em **Done**.

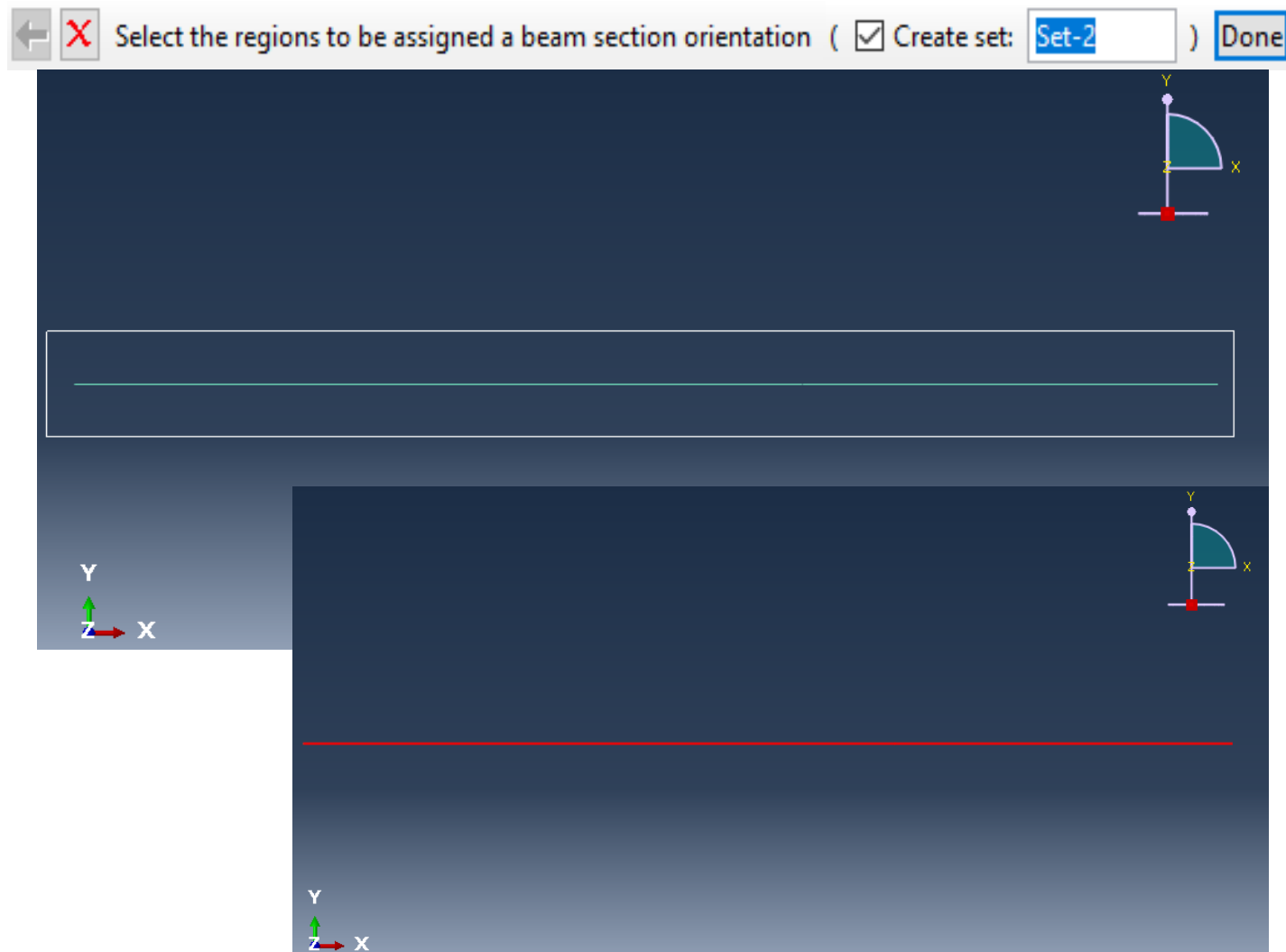
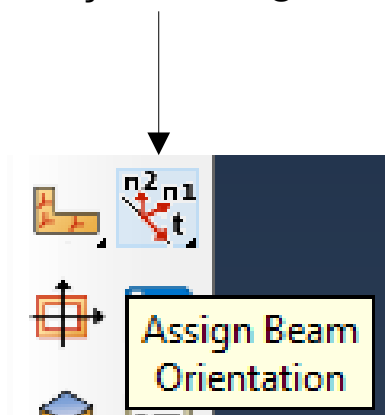




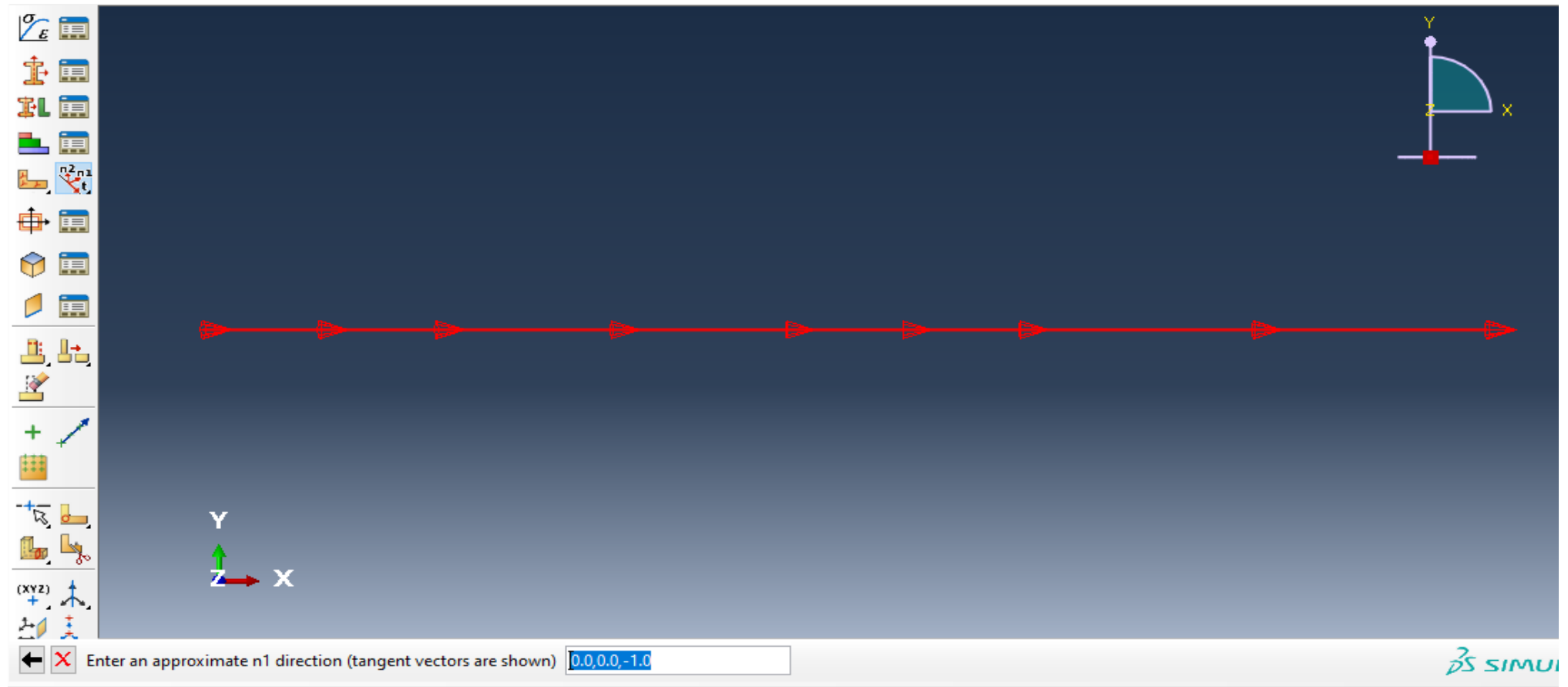
Passo 9

Selecionar o modelo para definir a orientação.

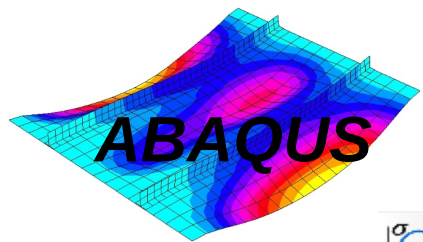
Clique aqui para definir a orientação da seção da viga.



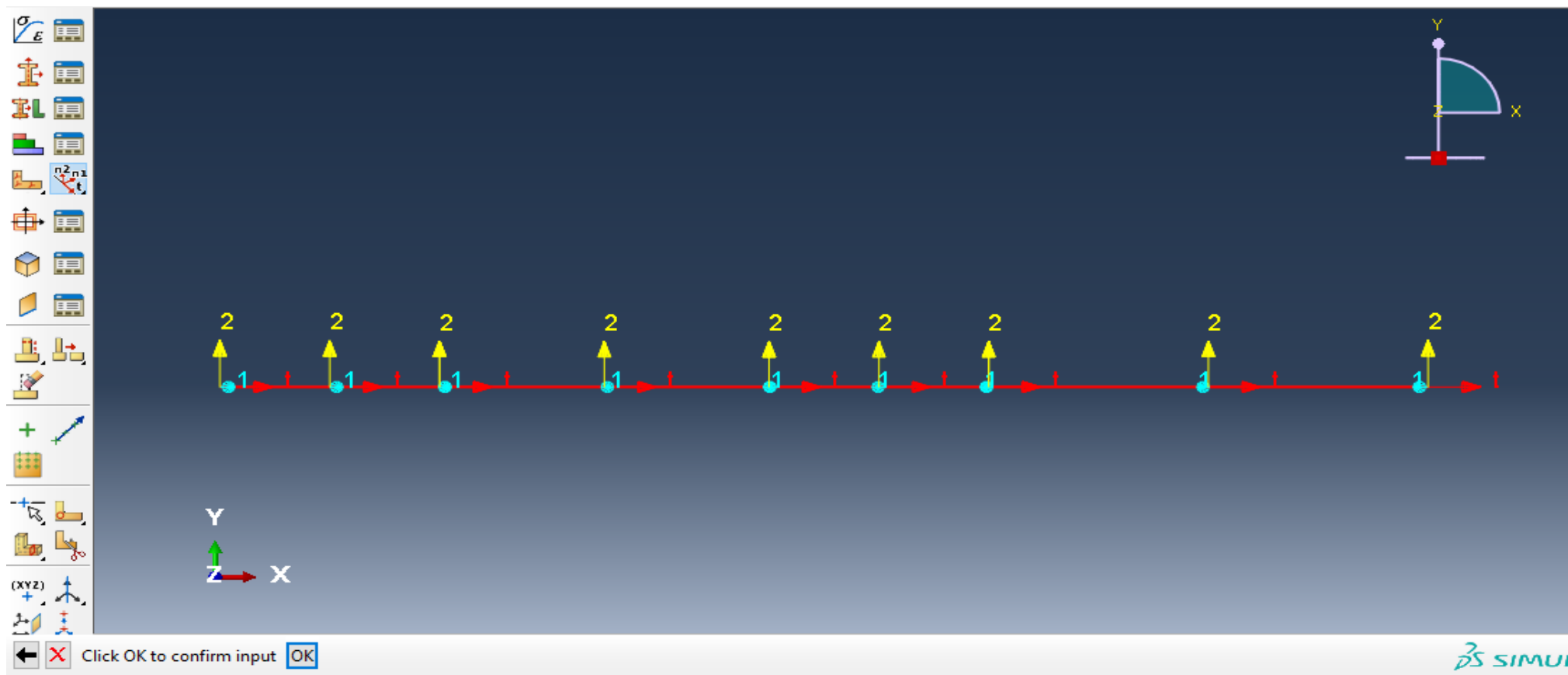
Em seguida, clique em **Done**.



Aceite está definição e
pressione Enter.

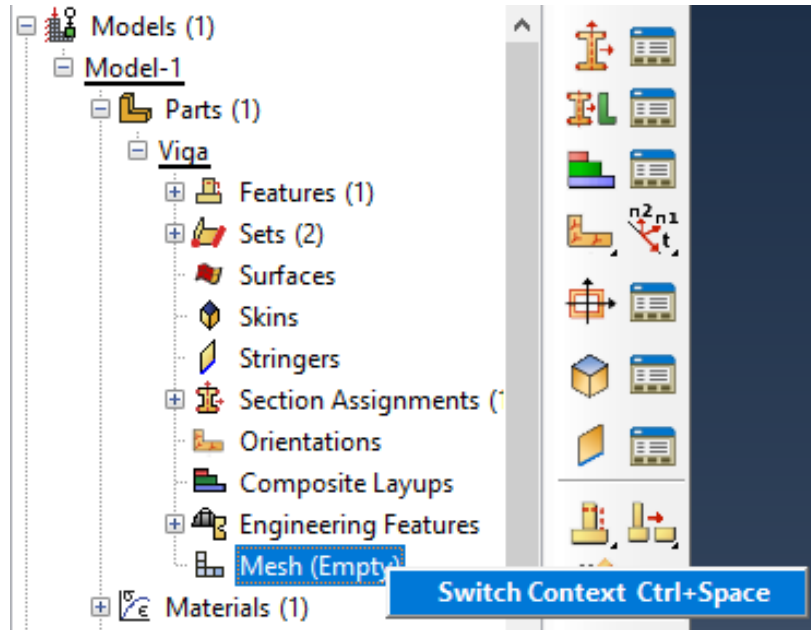


Passo 9

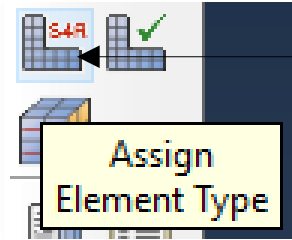


Clique em Ok, depois
pressione Esc.

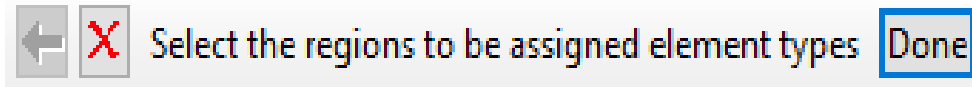
Passo 10



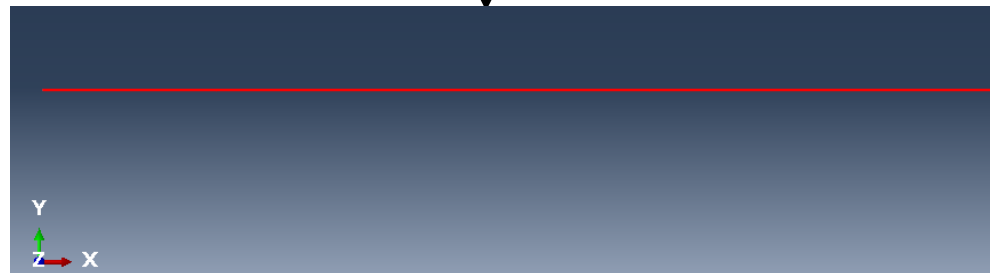
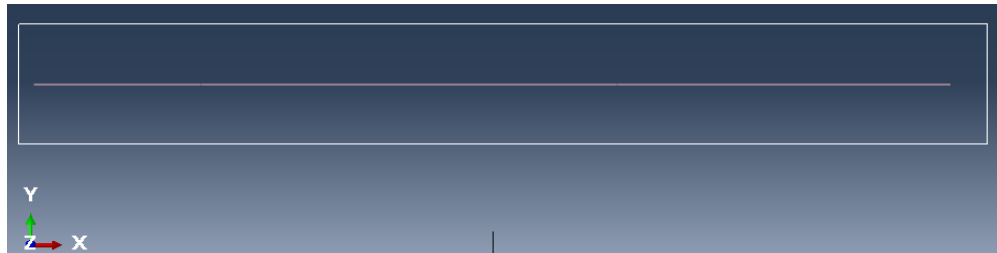
Clique com o botão direito sobre **Mesh** e, em seguida, clique sobre **Switch Context** para habilitar a barra de controle de malha (mesh)

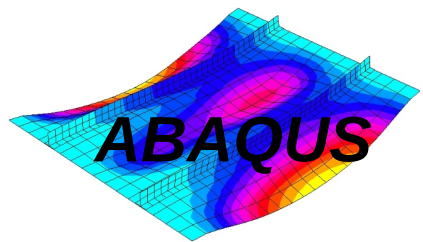


Clique aqui para definir o elemento finito de viga.



Em seguida, clique em **Done**.





Passo 10

Element Type

Element Library
☒ Standard ☐ Explicit

Geometric Order
☒ Linear ☐ Quadratic

Family
Acoustic
Beam
Coupled Temperature-Displacement
Gasket

Line

☐ Hybrid formulation

Element Controls

Beam type: ☒ Shear-flexible ☐ Cubic formulation

Scaling factors: Linear bulk viscosity: 1

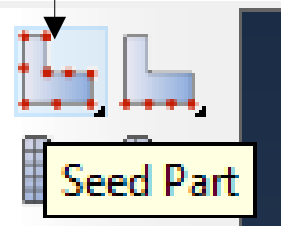
B21: A 2-node linear beam in a plane.

Note: To select an element shape for meshing, select "Mesh->Controls" from the main menu bar.

OK Defaults Cancel

Clique em Ok, depois pressione Esc.

Clique aqui para definir a posição e quantidade de nós da malha.



Global Seeds

Sizing Controls

Approximate global size:

☒ Curvature control
Maximum deviation factor ($0.0 < h/L < 1.0$):
(Approximate number of elements per circle: 8)

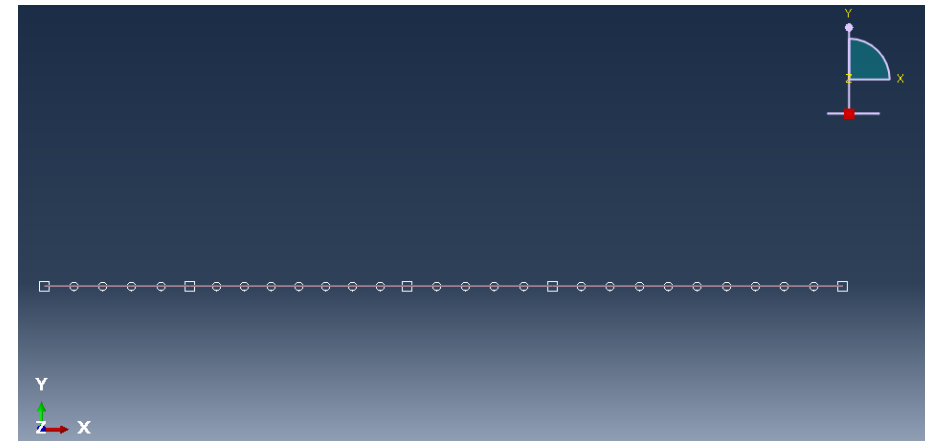
Minimum size control

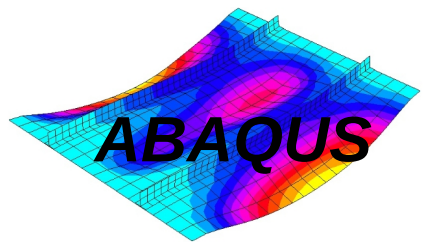
☒ By fraction of global size ($0.0 < \text{min} < 1.0$)

☐ By absolute value ($0.0 < \text{min} < \text{global size}$)

OK Apply Defaults Cancel

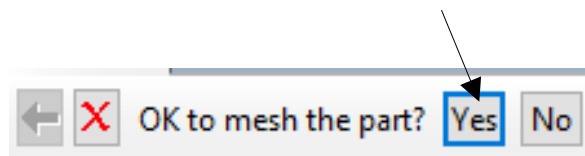
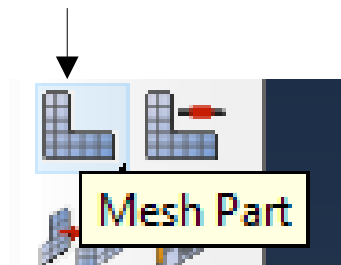
Clique em Apply e depois em Ok.

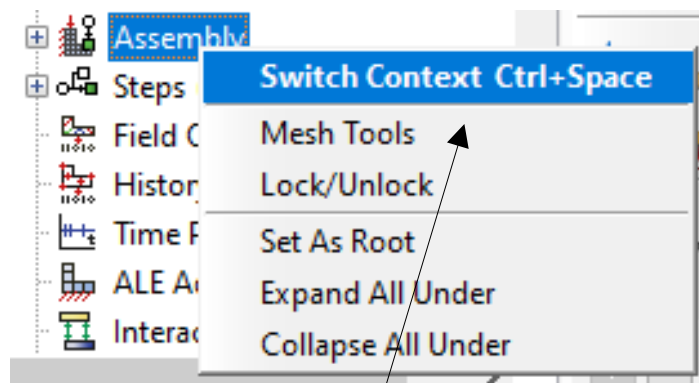




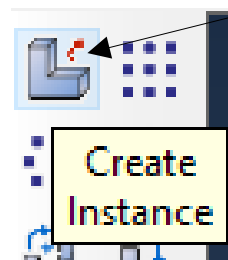
Passo 11

Clique aqui para gerar a malha.



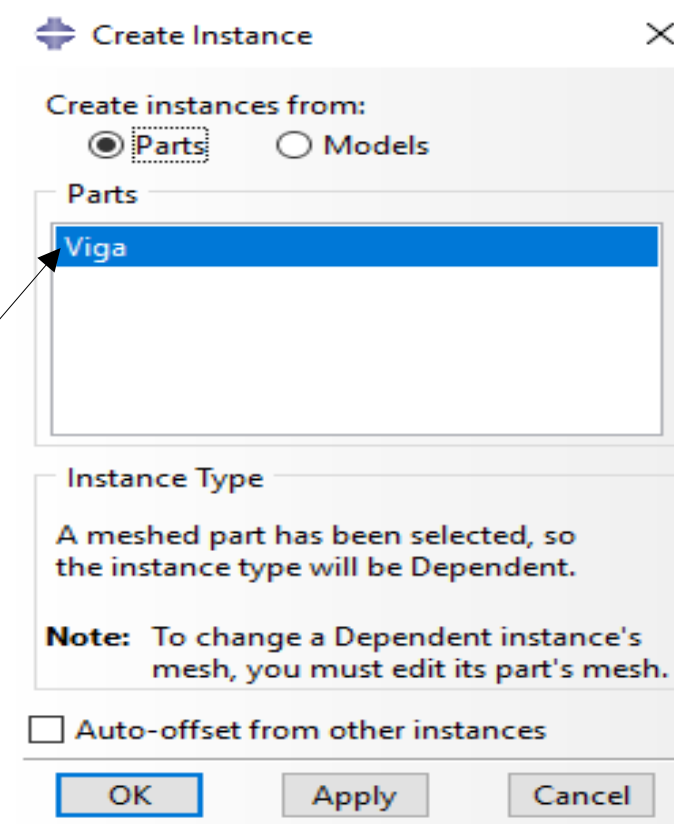


Na árvore do modelo clique com o botão direito sobre **Assembly** e, em seguida, clique sobre **Switch Context** para habilitar a barra de edição do conjunto.



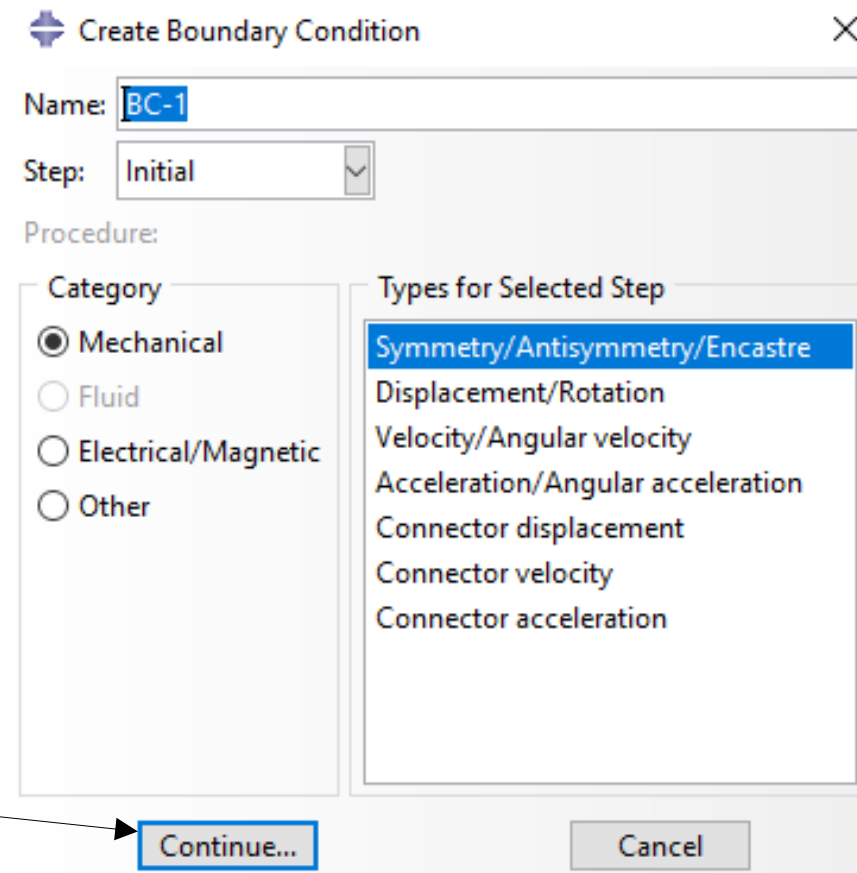
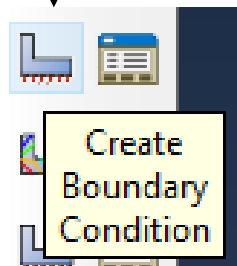
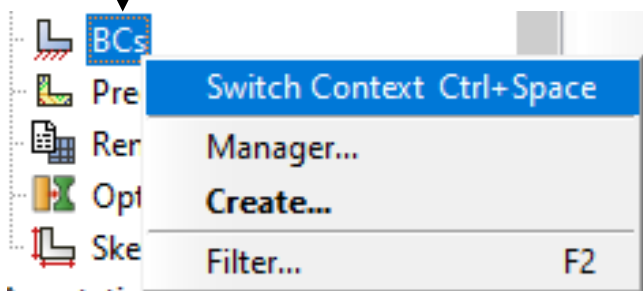
Selecione a Part que queira incluir no Assembly

Clique aqui para criar uma instancia do Part no conjunto (Assembly).

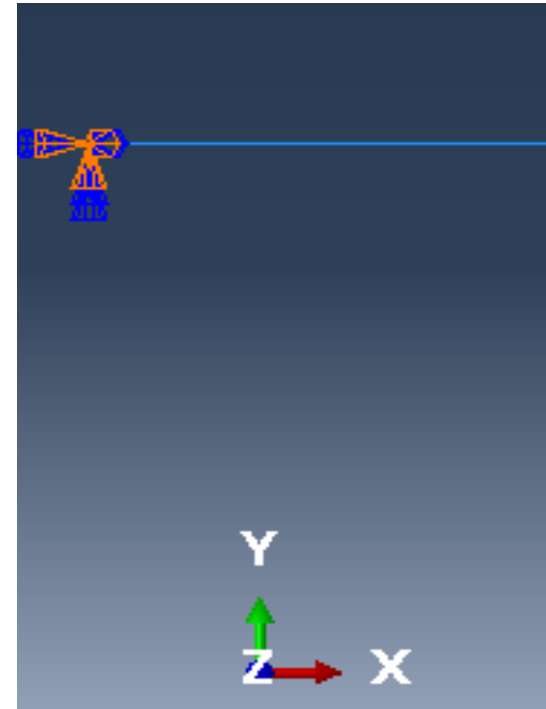
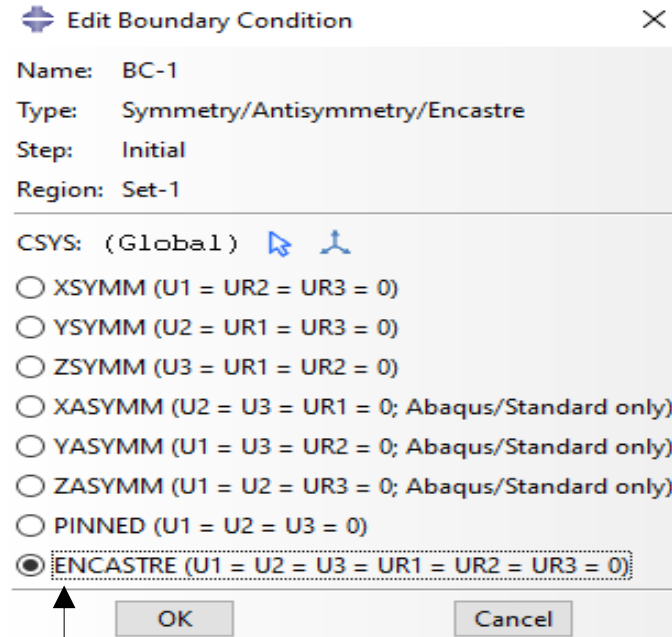
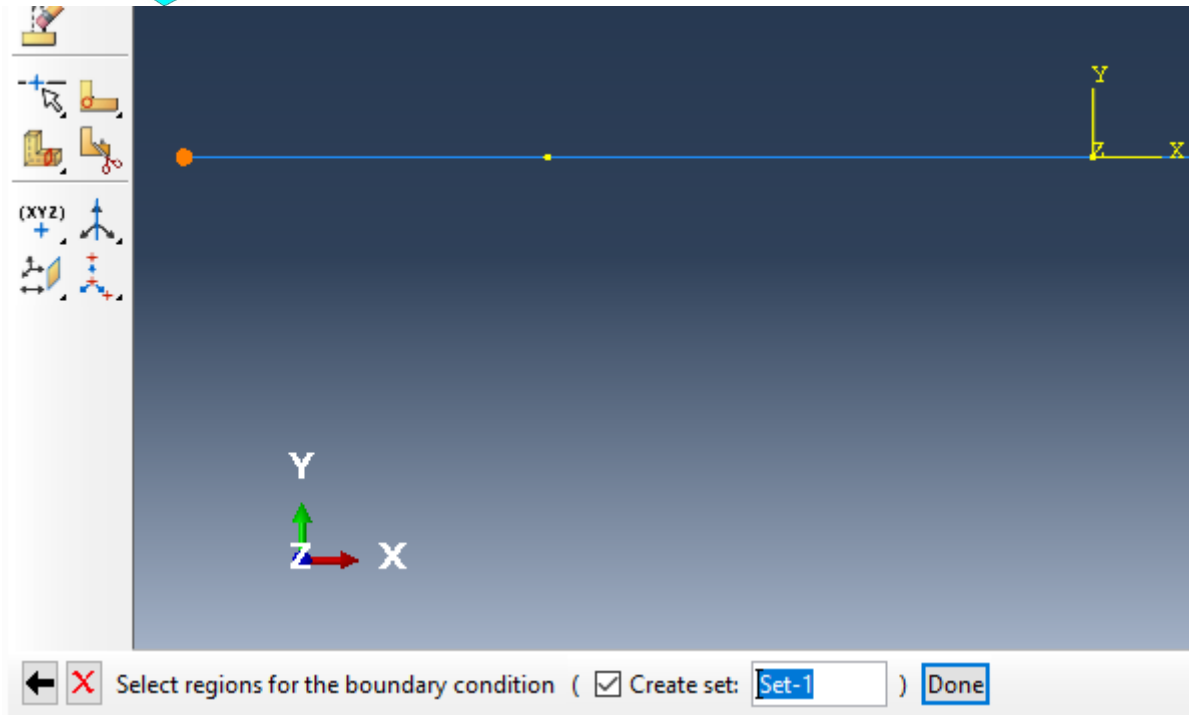


Na árvore do modelo clique com o botão direito sobre **BCs** e, em seguida, clique sobre **Switch Context** para habilitar a barra de condições de contorno.

Clique aqui para criar as condições de contorno.

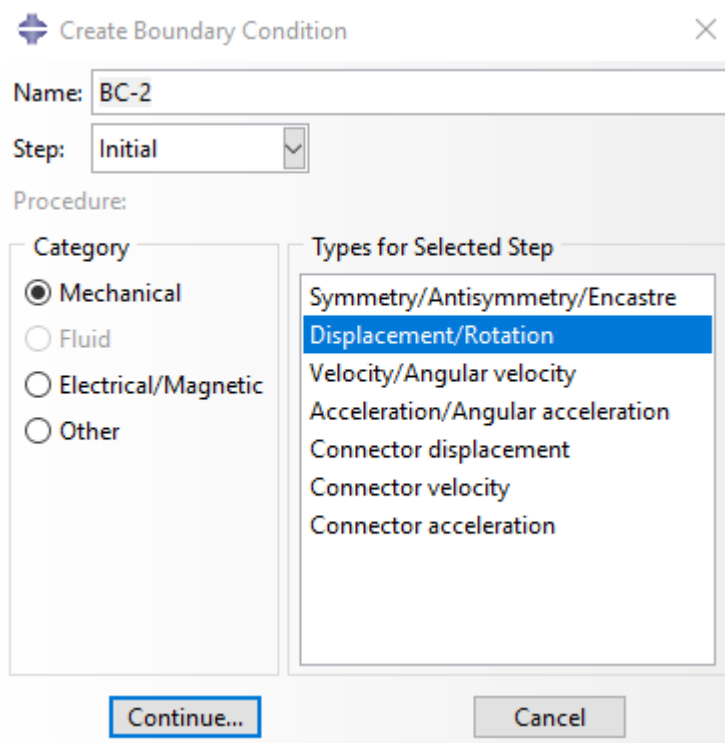


Clique em Continue...

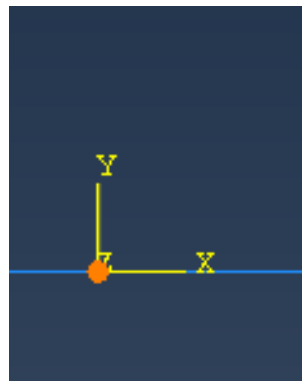


Selecione
ENCASTRE e em
seguida clique Ok.

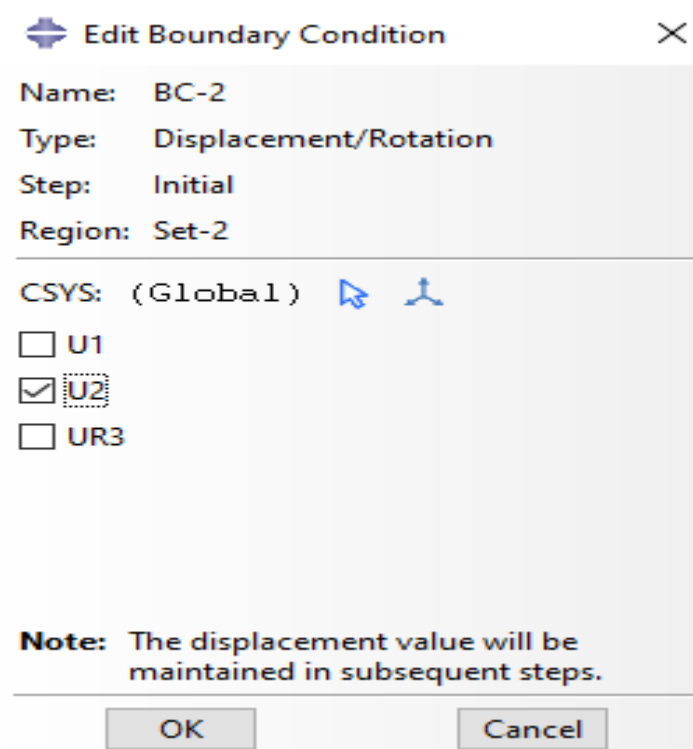
Repita a operação para a criação da condição de contorno, porém dessa vez selecione Displacement/Rotation.



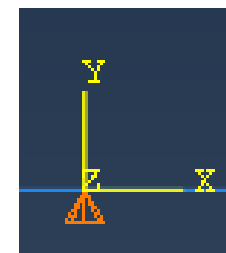
Clique em Continue...

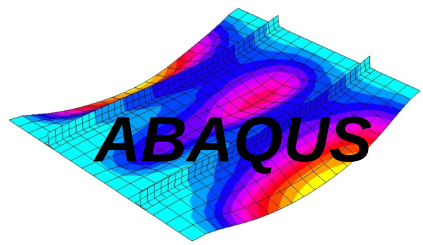


Selecione o ponto no meio da viga. Em seguida, clique em Done.

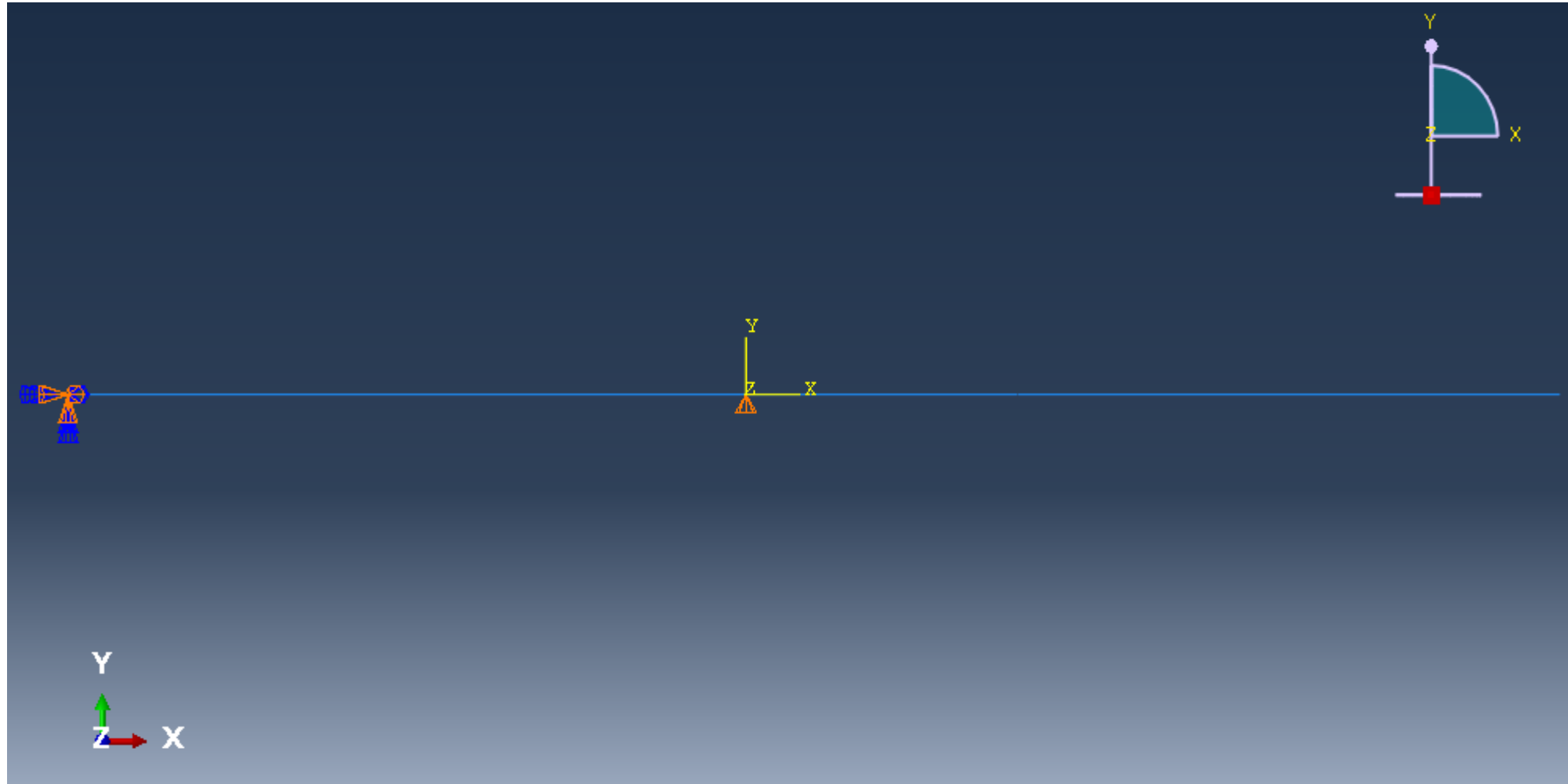


Selecione U2 para restringir o deslocamento na direção Y. Em seguida, clique Ok.

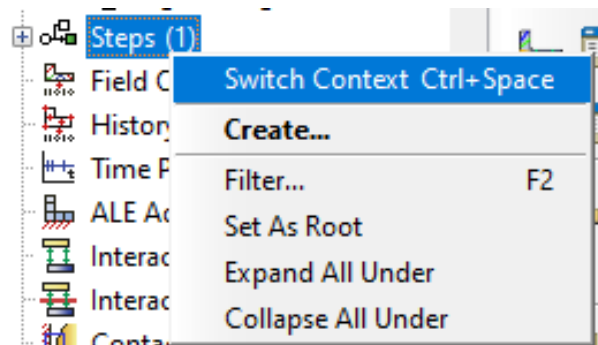




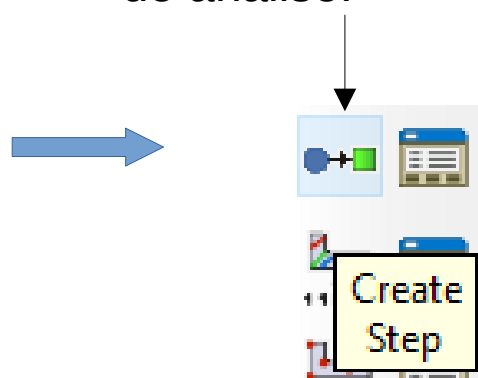
Passo 13



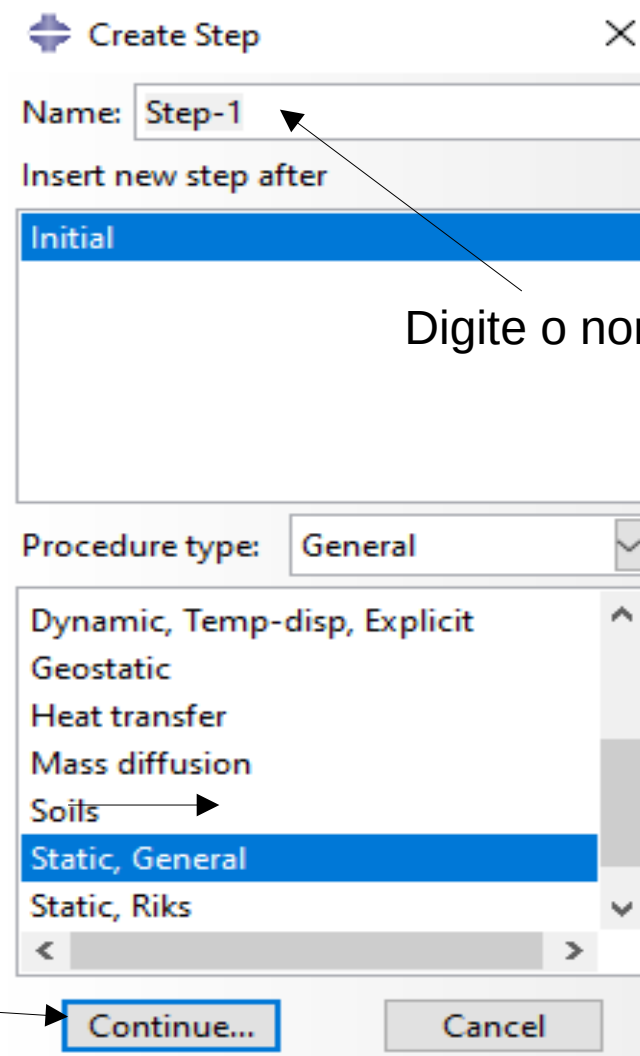
Na árvore do modelo clique com o botão direito sobre **Steps(1)** e, em seguida, clique sobre **Switch Context**.



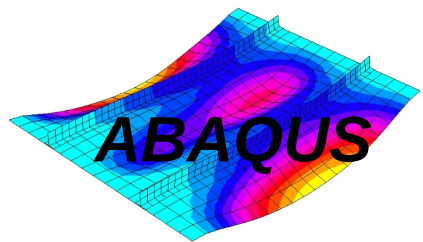
Clique aqui para criar um novo Step e definir o tipo de análise.



Clique em Continue...



Digite o nome do Step



Passo 14

Edit Step

Name: Step-1

Type: Static, General

Basic Incrementation Other

Description: Analise_estatica

Time period: 1

Nlgeom: ☒ Off (This setting controls the inclusion of nonlinear effects of large displacements and affects subsequent steps.)
☐ On

Automatic stabilization: None

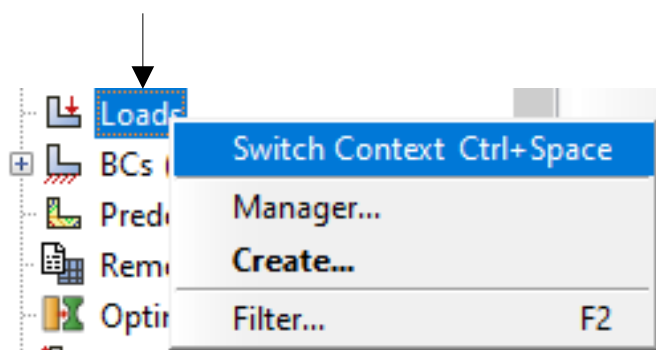
☐ Include adiabatic heating effects

OK Cancel

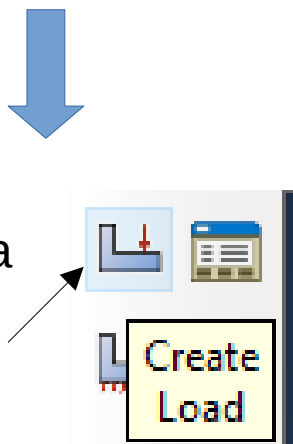
Digite uma descrição.

Termine clicando Ok

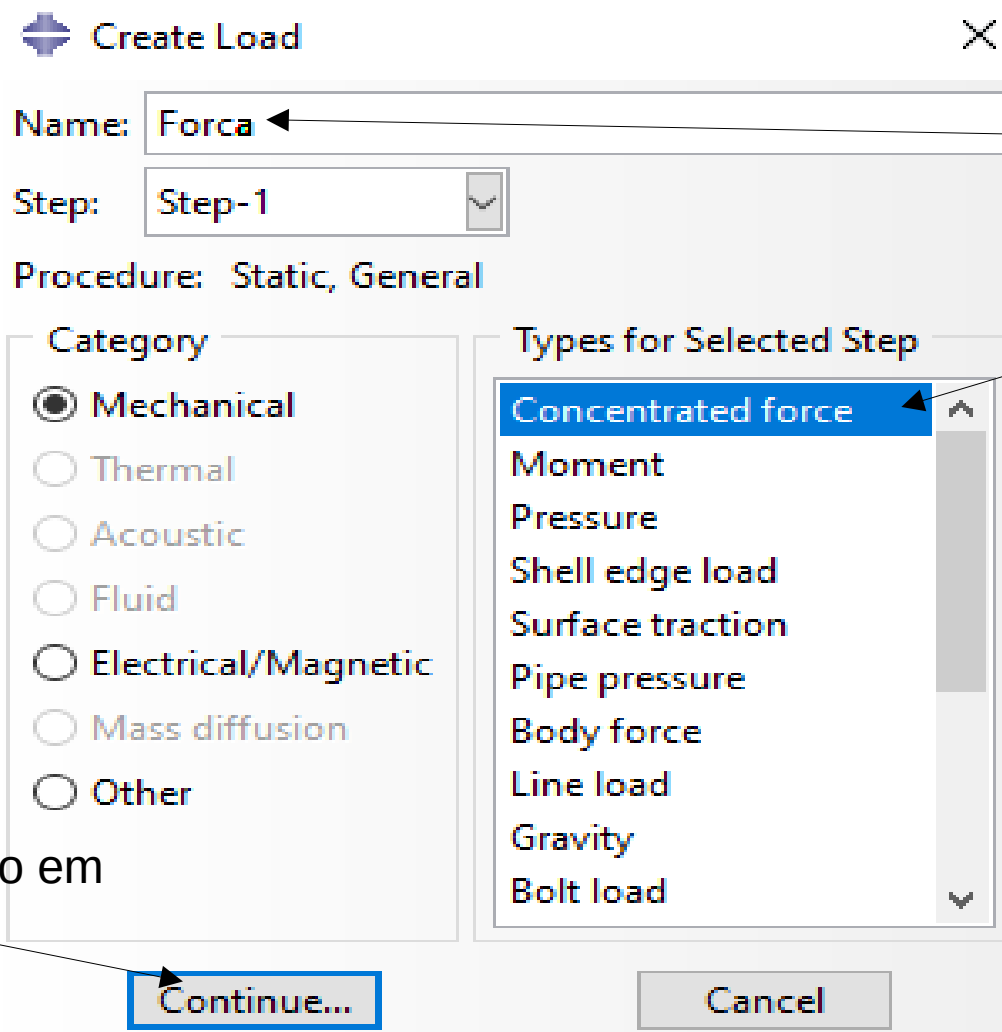
Na árvore do modelo clique com o botão direito sobre **Loads** e, em seguida, clique sobre **Switch Context**.



Clique aqui para criar um carregamento

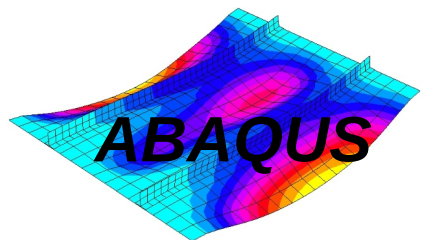


Termine clicando em Continue...

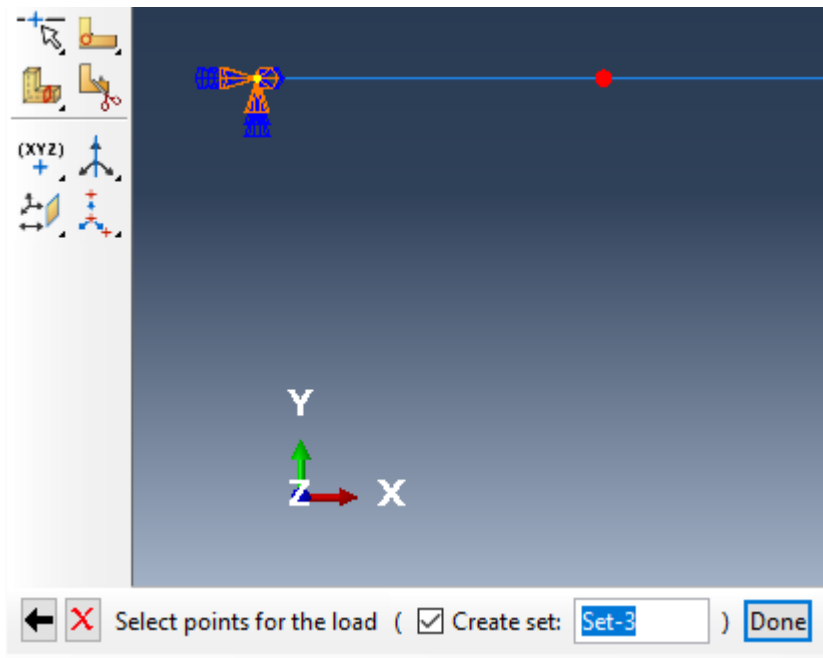


Digite o nome da força/carga.

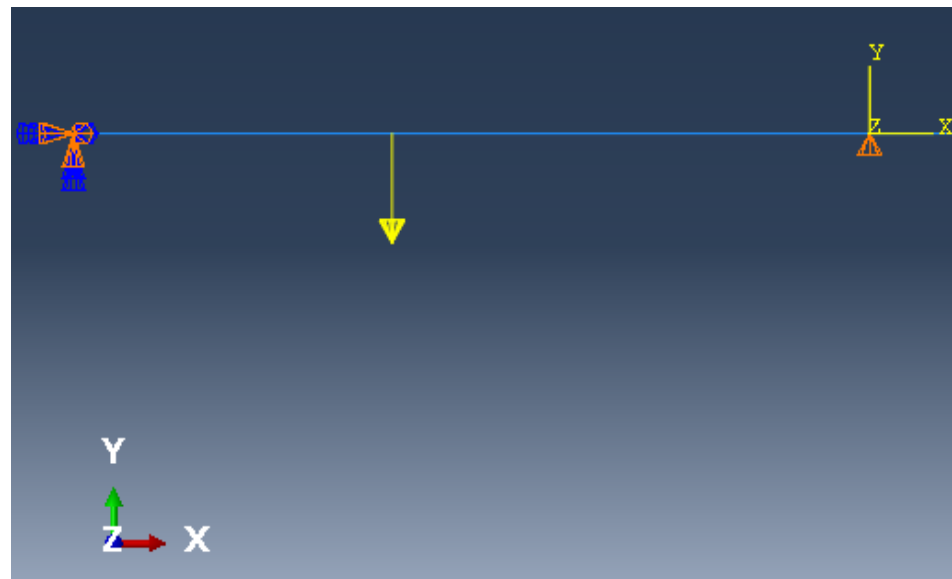
Selecione Concentrated Force.



Passo 15



Selecione o ponto em destaque e clique em Done.



Edit Load

Name: Força

Type: Concentrated force

Step: Step-1 (Static, General)

Region: Set-3

CSYS: (Global)

Distribution: Uniform $f(x)$

CF1: 0

CF2: -20000

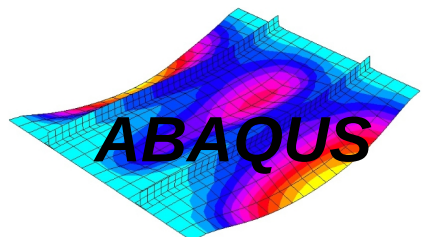
Amplitude: (Ramp)

☐ Follow nodal rotation

Note: Force will be applied per node.

OK Cancel

Atribua as definições conforme a imagem acima e clique em Ok.

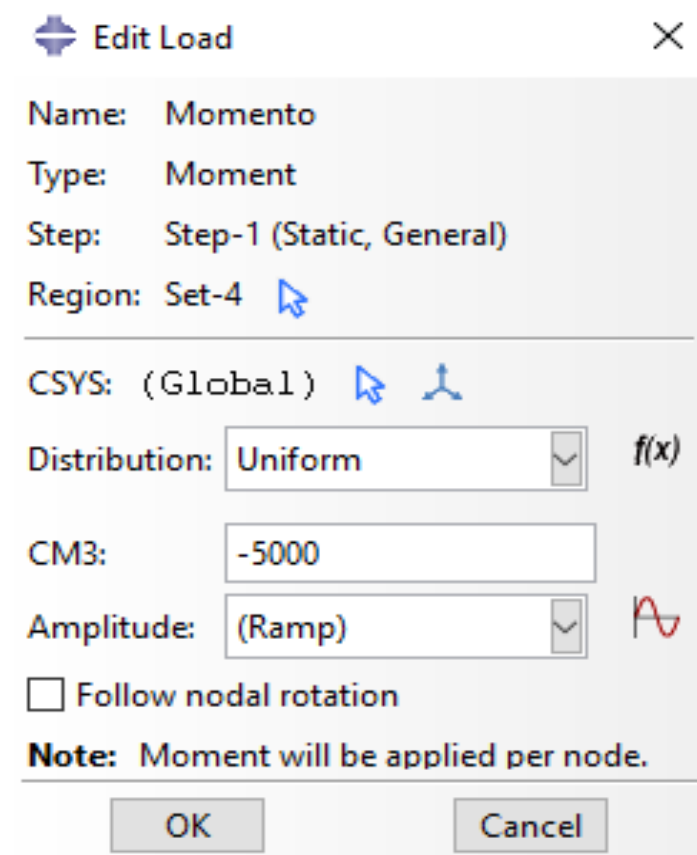
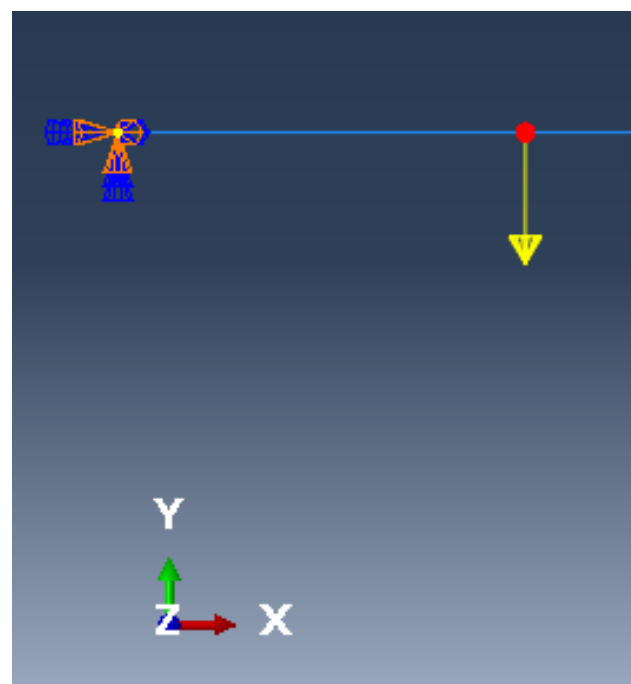
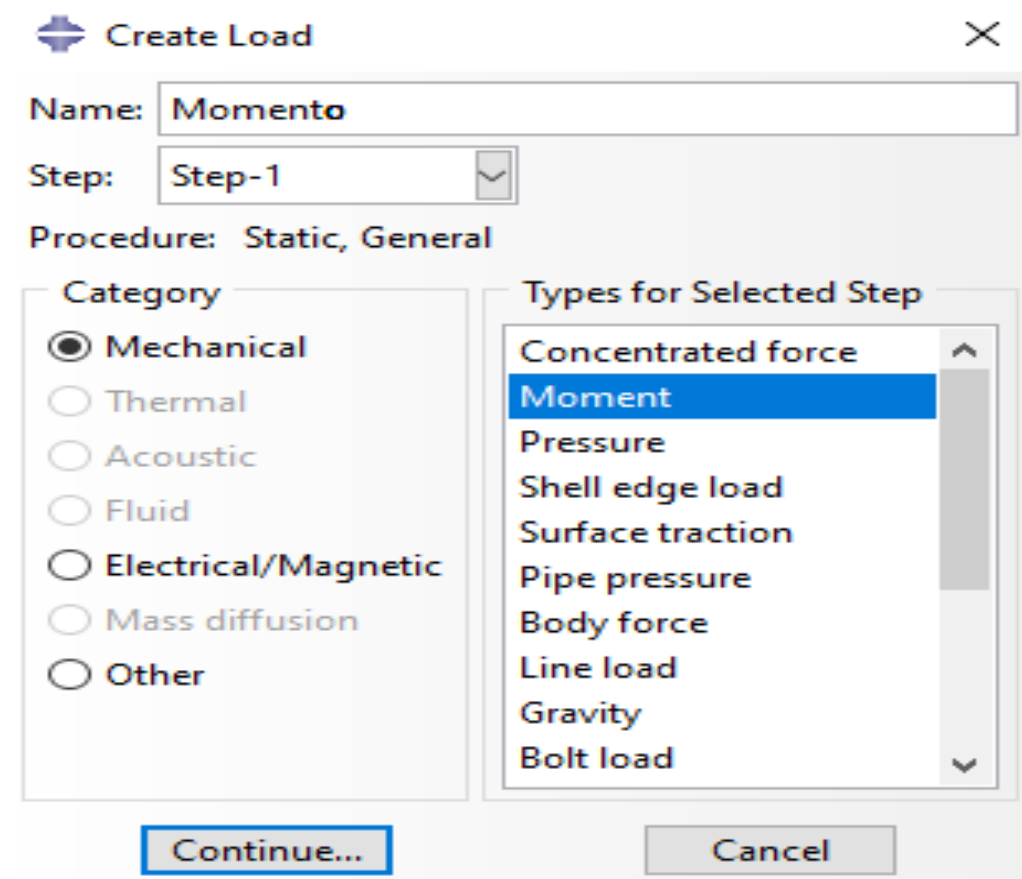


Passo 15

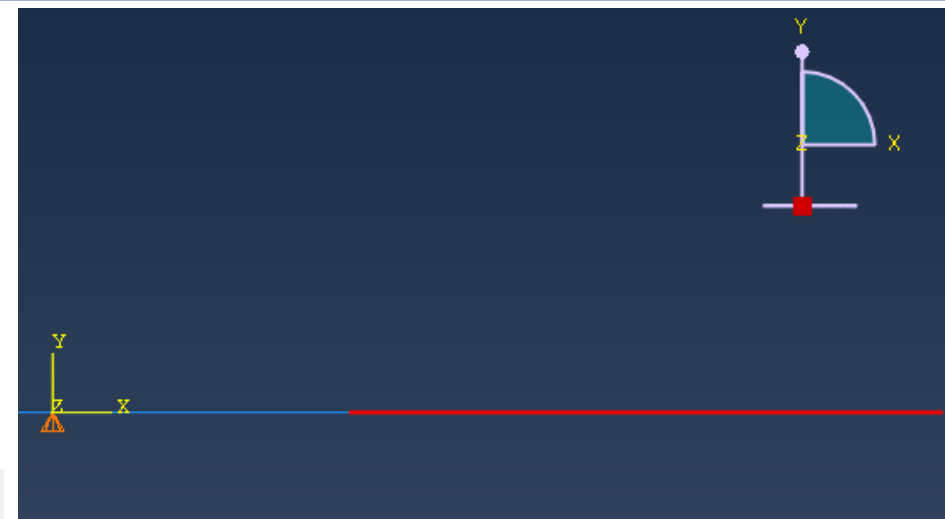
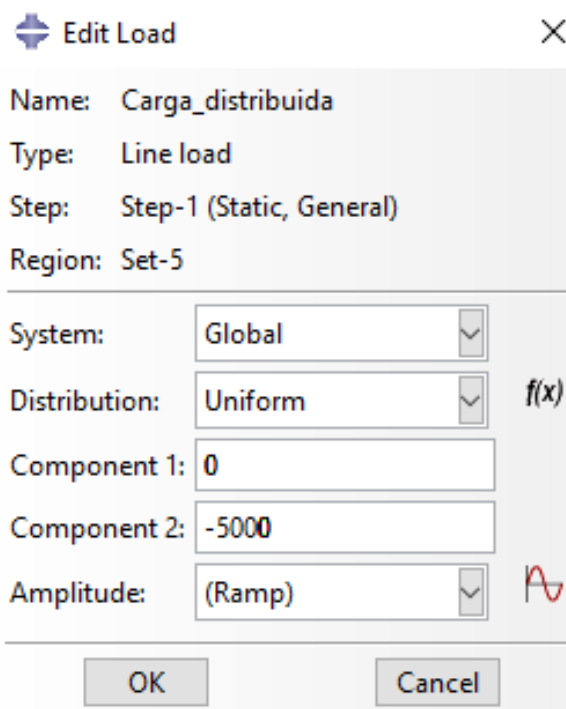
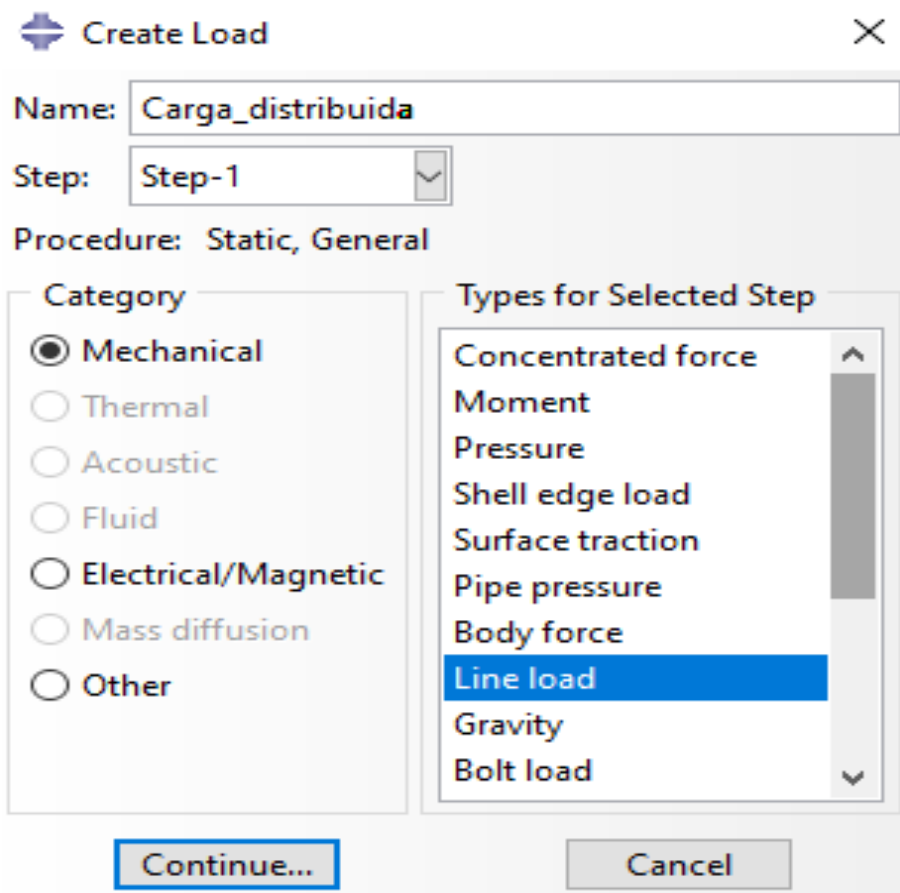
Repita a operação para a criação dos esforços, porém, dessa vez, selecione Moment. Em seguida, clique em Continue...

Selecione o mesmo ponto do caso anterior e clique em Done.

Atribua as definições conforme a imagem a esquerda e clique em Ok.

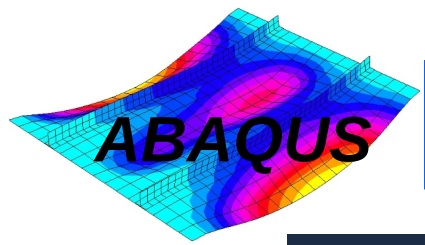


Repita a operação para a criação dos esforços, porém, dessa vez, selecione Line load. Em seguida, clique em Continue...

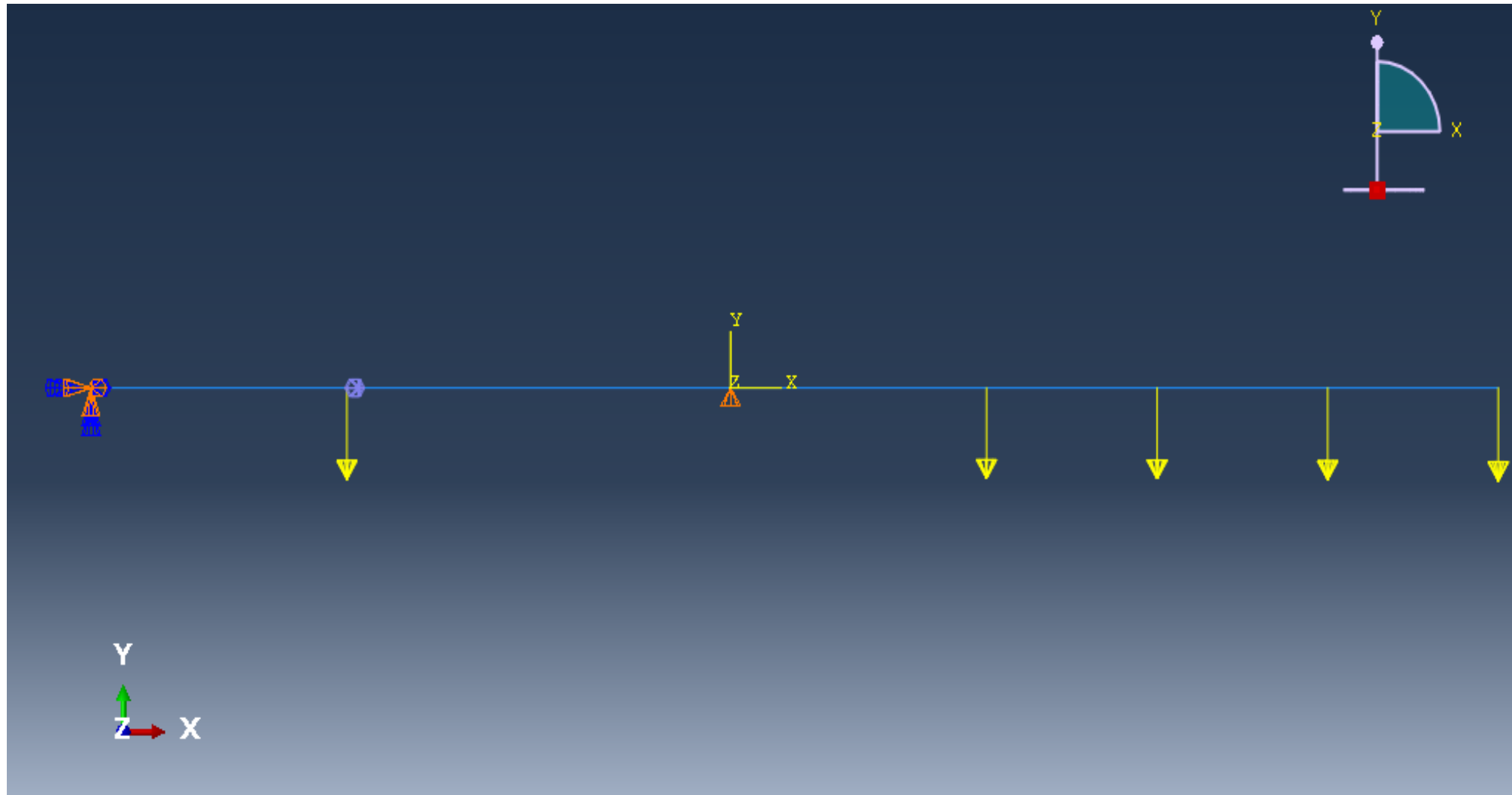


Selecione o segmento final do lado direito e clique em Done.

Atribua as definições conforme a imagem a esquerda e clique em Ok.

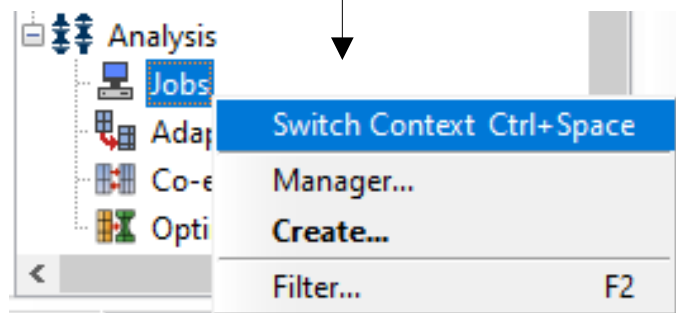


Passo 15

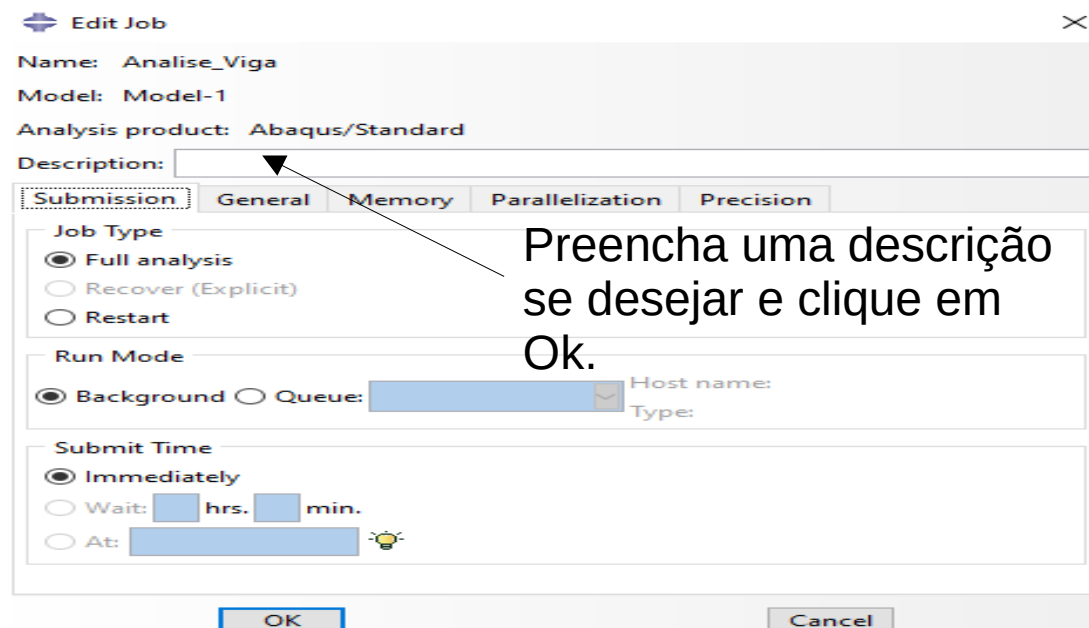
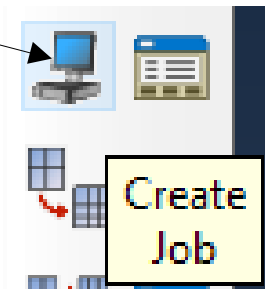


Passo 16

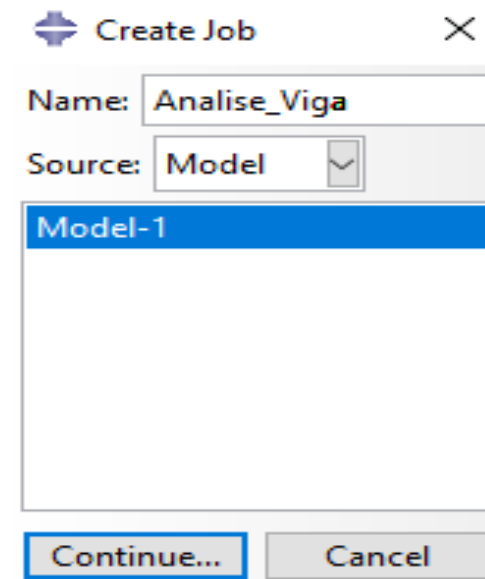
Na árvore do modelo clique com o botão direito sobre **Jobs** e, em seguida, clique sobre **Switch Context**.



Clique aqui para criar um uma execução de análise (Job).



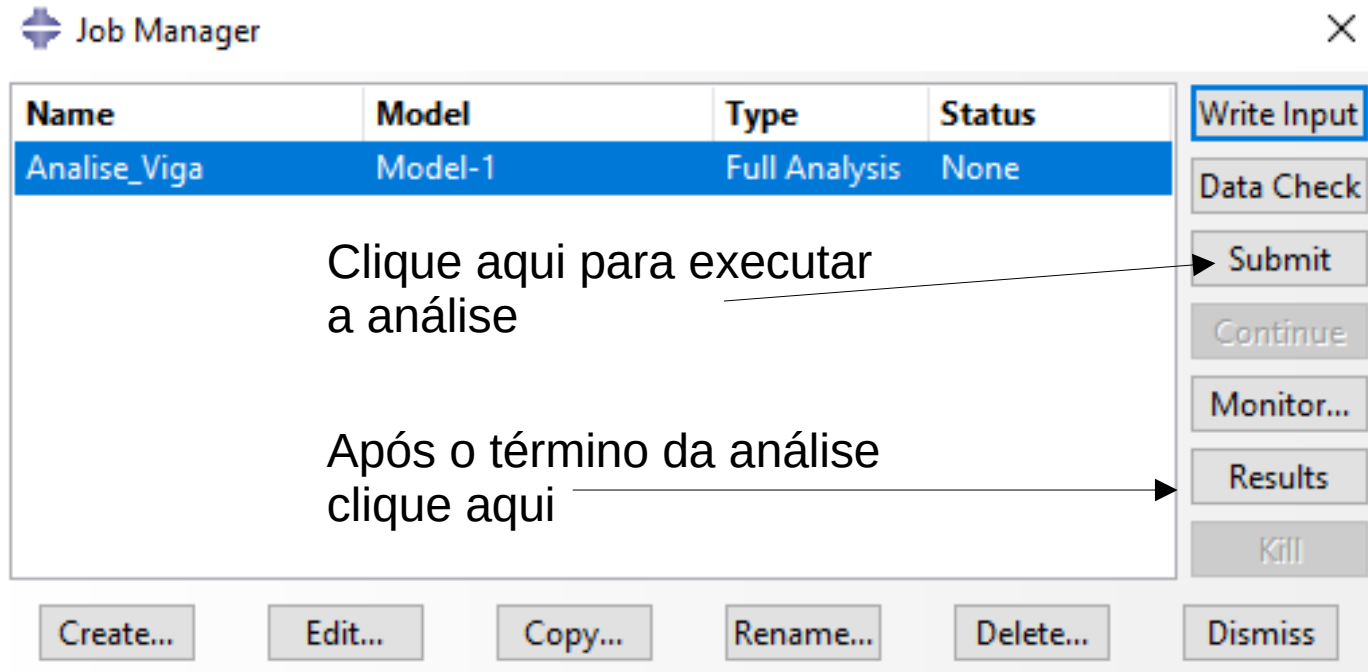
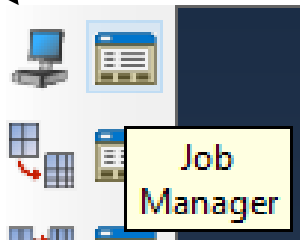
Preencha uma descrição se desejar e clique em Ok.



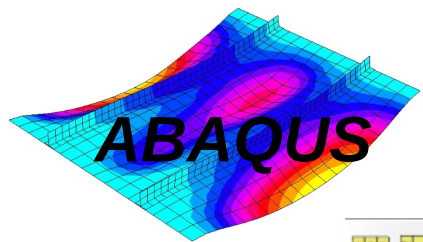
Atribua as definições conforme a imagem acima e clique em Continue...

Passo 17

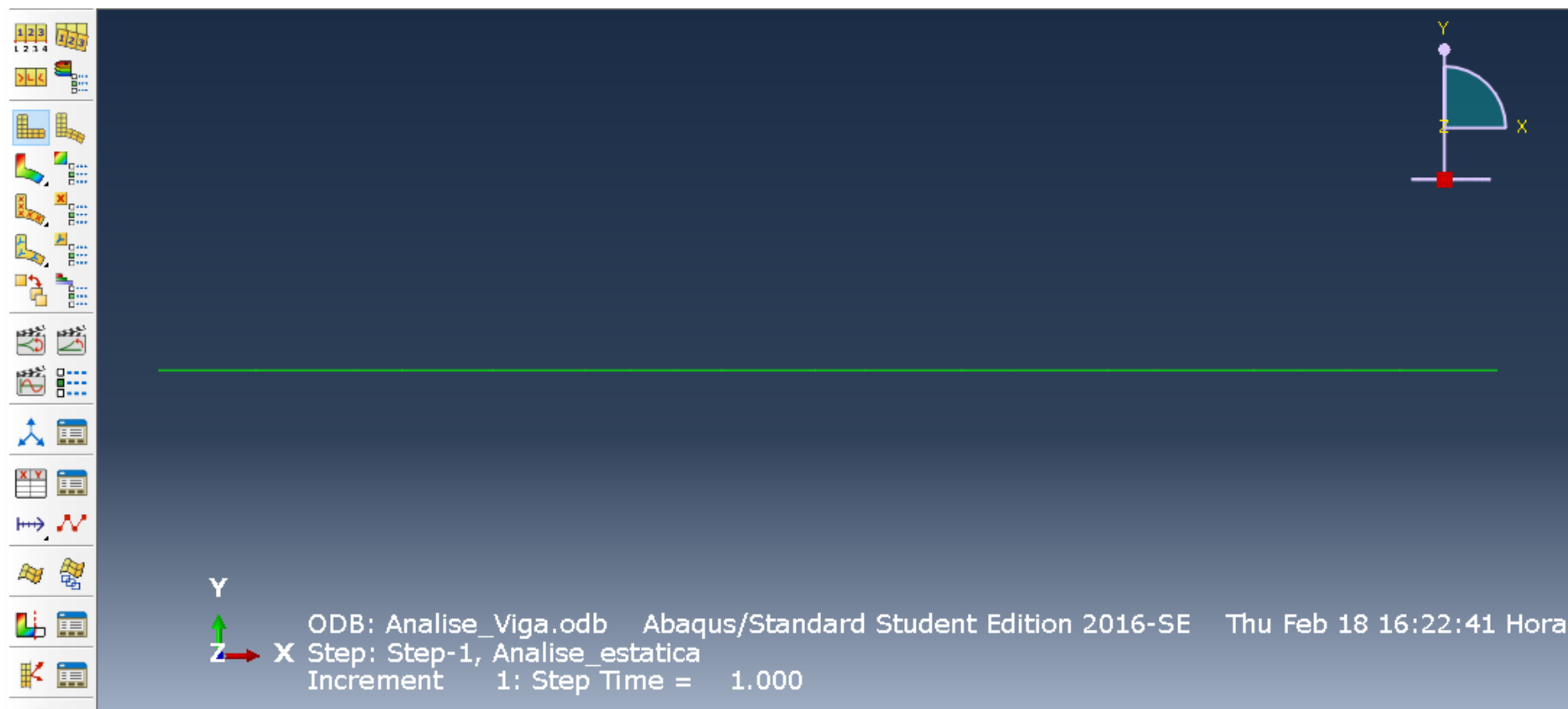
Clique aqui para abrir a janela de gerenciamento da execução de análise.



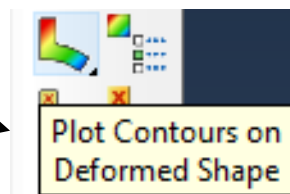
The job input file "Analise_Viga.inp" has been submitted for analysis.
 Job Analise_Viga: Analysis Input File Processor completed successfully.
 Job Analise_Viga: Abaqus/Standard completed successfully.
 Job Analise_Viga completed successfully.

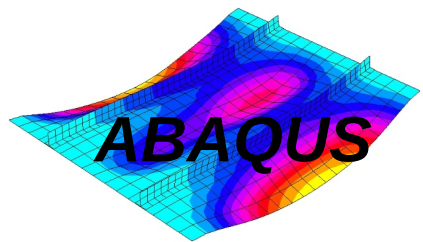


Passo 18



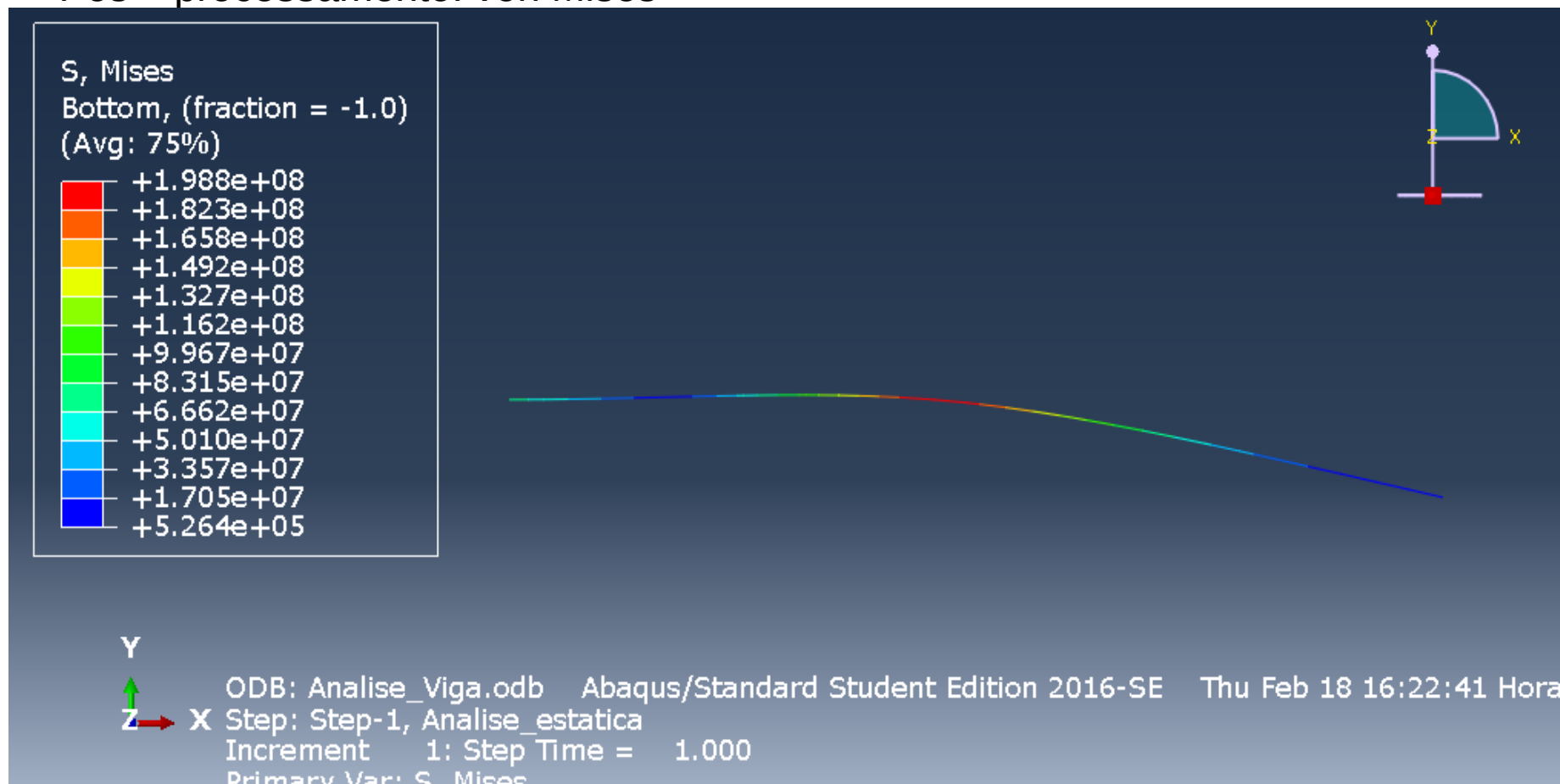
Clique aqui para abrir os resultados da análise.

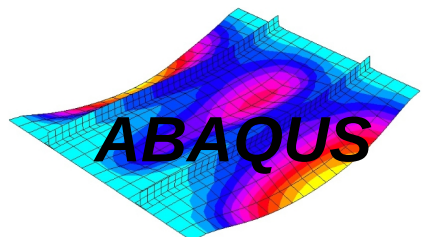




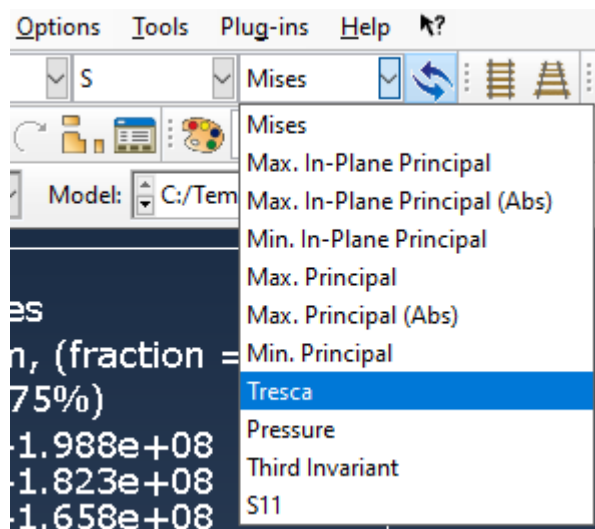
Passo 18

Pós – processamento: von Mises





Passo 18



Pós – processamento:
Tresca

