How to Set 3D Solid Stress Boundary Conditions to Mimic Line Element Beam Theory Behavior

Andrew Winter

1 Introduction

When developing a finite element method (FEM) model from scratch, it is often necessary to validate model results such as deflections and stresses against known solutions derived from a line element beam theory such as Euler-Bernoulli or Timoshenko-Ehrenfest (i.e. shear deformable). In order to perform such a validation accurately, it is necessary to set the boundary conditions (BCs) correctly to avoid causing stress concentrations or over-restraining rotations and deflections. This article demonstrates how to properly setup such BCs when performing 3D Solid Stress analysis in Simcenter STAR-CCM+ using a simple beam structure example.

2 Example Structure

A cantilevered beam with a rectangular cross-section subjected to a uniform distributed load was selected as the example structure. Fig. 1 shows the beam geometry created using 3D-CAD within STAR-CCM+ and Table 1 lists the selected beam parameters.

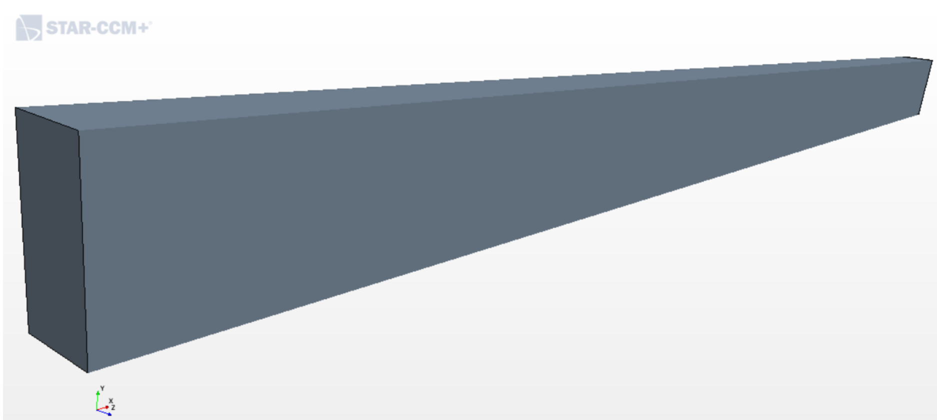


Figure 1: Cantilever Beam Geometry

Table 1: Beam Parameters

Dimensions		Material Properties	
Length, L	8.0 m	Density, ρ	7850 kg/m^3
Depth, h	0.5 m	Elasticity, E	207 GPa
Width, b	0.25 m	Poisson's Ratio, ν	0.3

A Directed Mesh of uniformly-sized elements was generated to model the beam, where the fixed and free ends of the beam were selected as the Source Surfaces and Target Surfaces. Additionally, the physics models displayed in Fig. 2 were selected to setup the simulation.

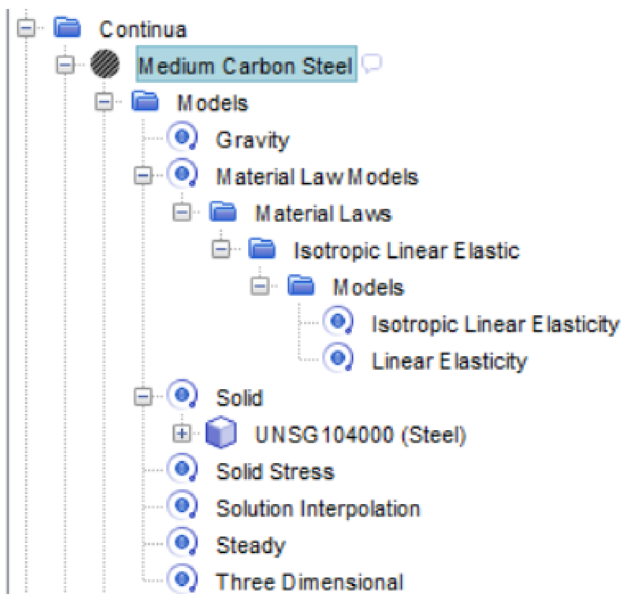


Figure 2: Physics Models

3 STAR-CCM+ Boundary Conditions

In order to specify boundary conditions and apply external loads in a Solid Stress simulation, Segments need to be created in the model Region. There are three types of Segments available in STAR-CCM+ including Point, Curve, and Surface Segments as described in the Segments (Loads and Constraints) Reference documentation.

To define a Segment, expand the Regions part of the simulation tree as well as the name of the Region in which the Segment is to be defined. Right-click on Segments to select Create Segment $\longrightarrow$ [Type] Segment, where [Type] can be Point, Curve, or Surface as shown in Fig. 3. For comparison purposes, segments were created for two different BCs, a fully-fixed end and a line element fixed end.

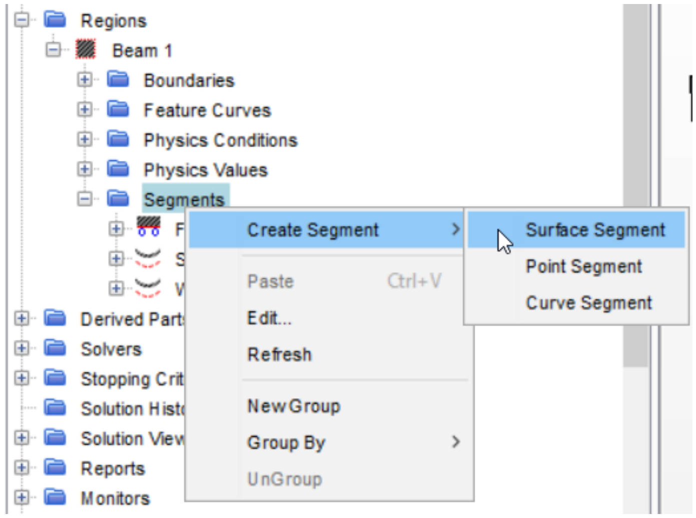
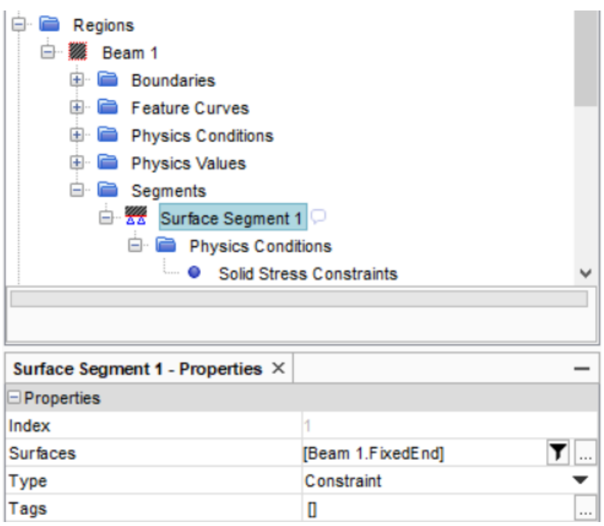


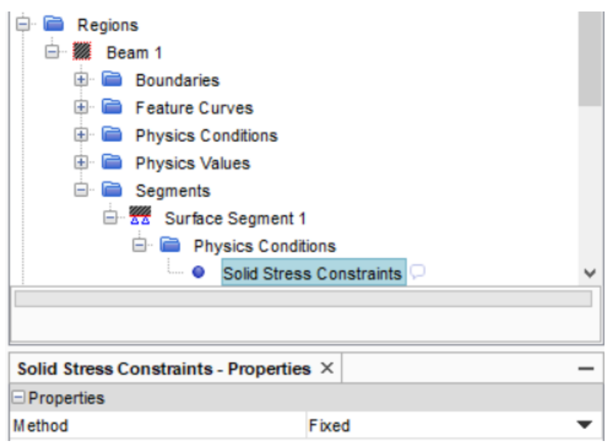
Figure 3: Segment Creation

3.1 Fully-Fixed End BC

This BC prevents displacement in any axis direction over the entire fixed end. Create an empty Surface Segment in the beam Region. Select the Segment, change its Type to Constraint, and select the fixed end in the Surfaces list as shown in Fig. 4a. Expand the Segment and its Physics conditions tree to select Solid Stress Constraints and verify that the Method is set to Fixed as shown in Fig. 4b.



(a) Segment



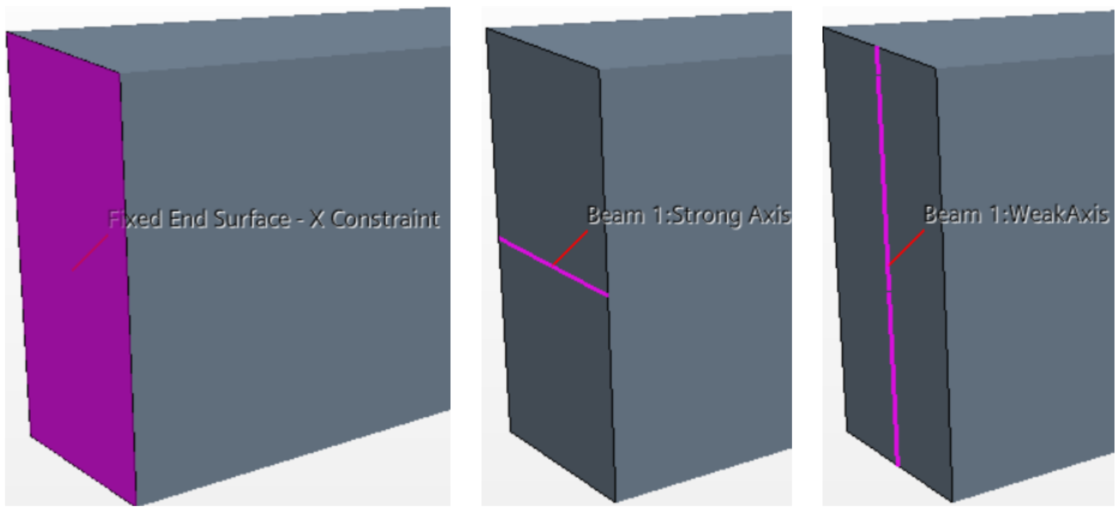
(b) Solid Stress Constraint

Figure 4: Fixed End Condition Properties

3.2 Line Element Fixed End BC

To facilitate line element fixed end BCs in a 3D FEM model, the fixed end surface must be split horizontally and vertically along the neutral axes for strong (z-axis) and weak (y-axis)

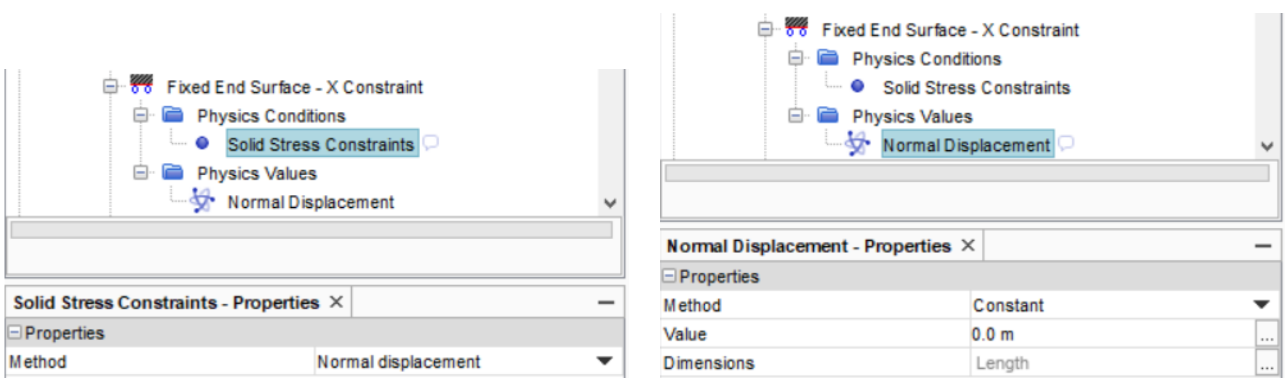
axis bending, respectively, as shown in Fig. 5.



(a) Normal Displacement (b) Strong Axis (c) Weak Axis

Figure 5: Beam Cross-Section Surface & Curve Segments

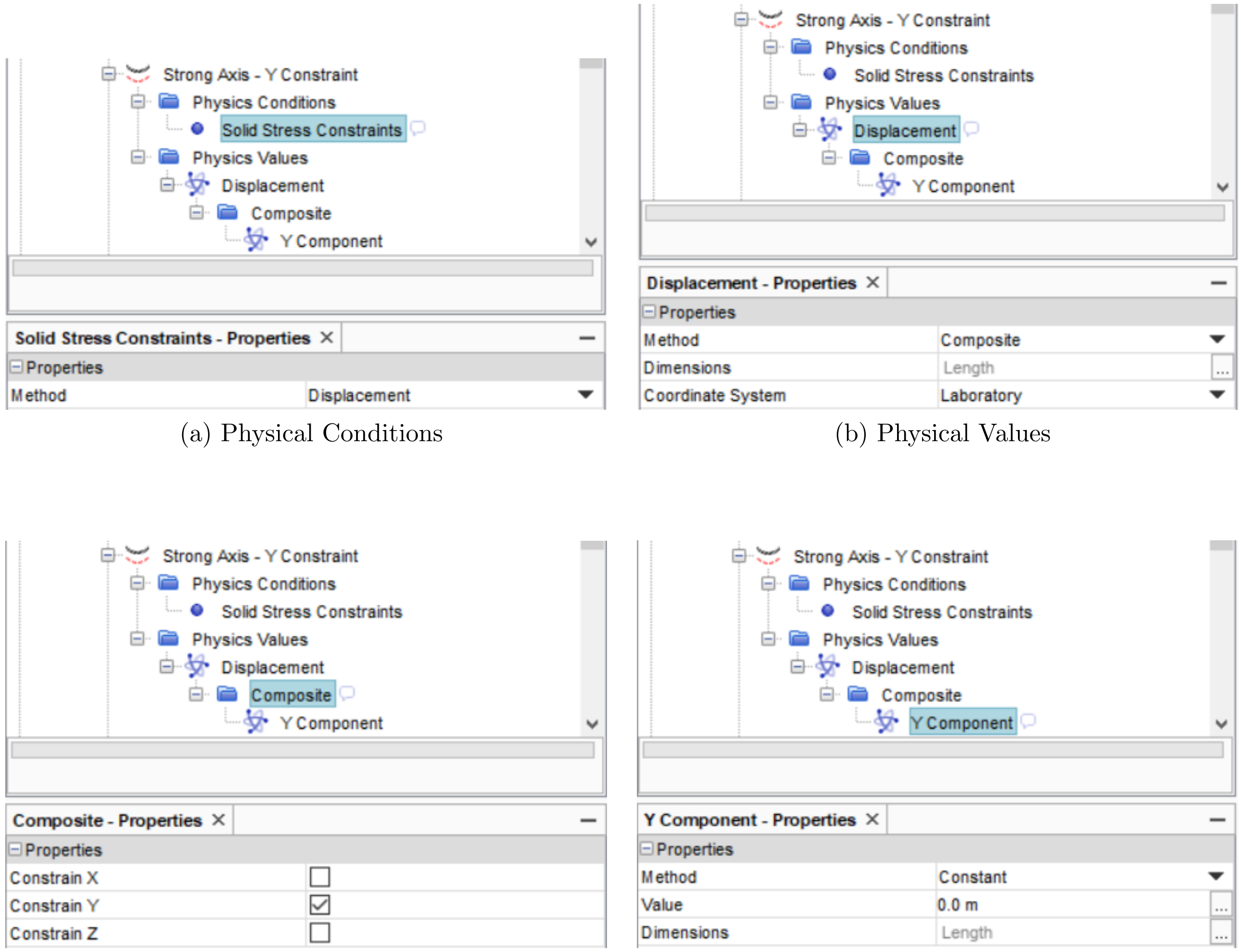
A Surface Segment must be created using the fixed end and Constraint as the Surfaces and Type. Normal Displacement should be specified as the Method for the Physics Conditions $\longrightarrow$ Solid Stress Constraint, whereas Constant and 0.0 m should be specified as the Method and Value for the Physics Values $\longrightarrow$ Normal Displacement as shown in Fig. 6.



(a) Physics Conditions (b) Physics Values

Figure 6: Normal Displacement-Only Surface Segment Definition

Following the strong axis example given in Fig. 7, Curve Segments should be created for the strong and weak axis curves. Displacement should be selected for the Solid Stress Constraints. Only a single displacement component should be restrained for each segment, which requires selecting Composite as the Method for Displacement. Next, click on Composite and check the Constrain Y and Constrain Z boxes for the strong and weak axis Curve Segments, respectively. Finally, set the Method and Value to Constant and 0.0 m for each Component under Composite.



(a) Physical Conditions

(b) Physical Values

(c) Composite Method

(d) Y-Component

Figure 7: Curve Segment Definition Example: Strong Axis

4 Comparing Results

To assess the accuracy of the STAR-CCM+ BCs, numerical and analytical maximum bending stresses should be compared at the fixed end. The Euler-Bernoulli analytical result is given below, where the second moment of area is $I = bh^3/12 = (0.25\text{ m})(0.5\text{ m})^3/12 = 2.604(10^{-3})\text{ m}^4$ and the unit weight per length is $w = \rho g A = (7850\text{ kg/m}^3)(9.81\text{ m/s}^2)(0.125\text{ m}^2) = 9626.06\text{ N/m} = 9.626\text{ kN/m}$.

$$\text{Stress Field}: \ \sigma_{xx}(x,y) = -\frac{M(x)y}{I} = \frac{ywL^2}{2I}\left[\left(\frac{x}{L}\right)^2 - 2\left(\frac{x}{L}\right) + 1\right]$$

$$\text{Min/Max Stress}: \ \sigma_{xx}(0,\pm h/2) = \mp\frac{whL^2}{4I} = \mp\frac{(9.626\text{ kN/m})(0.5\text{ m})(8\text{ m})^2}{4(2.604(10^{-3})\text{ m}^4)} = \mp 29571.26\text{ kPa}$$

Using a Maximum Report, the Solid Stress model gives differing values depending on the selected input part(s). Choosing the beam Region gives the maximum stress at an FEM element center, whereas choosing only the Boundaries produces an interpolated result at the beam's outer surface. A comparison of these results is provided in Fig. 8.

```
        Maximum of Stress Tensor[i,i] on Volume Mesh

Part        Value (MPa)
------      ------------
Beam 1      2.782560e+01
------      ------------
Total:      2.782560e+01
```

(a) Line Element BC: Beam Region

```
        Maximum of Stress Tensor[i,i] on Volume Mesh

Part                   Value (MPa)
-----------------      ------------
Beam 1: FreeSides      2.987754e+01
-----------------      ------------
           Total:      2.987754e+01
```

(b) Line Element BC: Beam Outer Surfaces

```
        Maximum of Stress Tensor[i,i] on Volume Mesh

Part        Value (MPa)
------      ------------
Beam 1      3.369639e+01
------      ------------
Total:      3.369639e+01
```

(c) Fully-Fixed BC: Beam Region

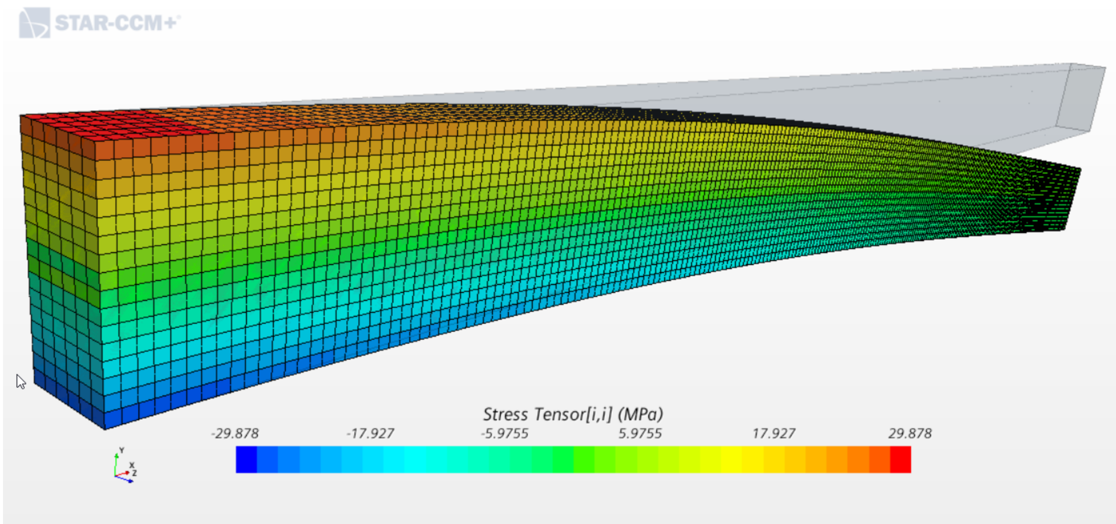
```
        Maximum of Stress Tensor[i,i] on Volume Mesh

Part                   Value (MPa)
-----------------      ------------
Beam 1: FreeSides      3.892387e+01
-----------------      ------------
           Total:      3.892387e+01
```

(d) Fully-Fixed BC: Beam Outer Surfaces

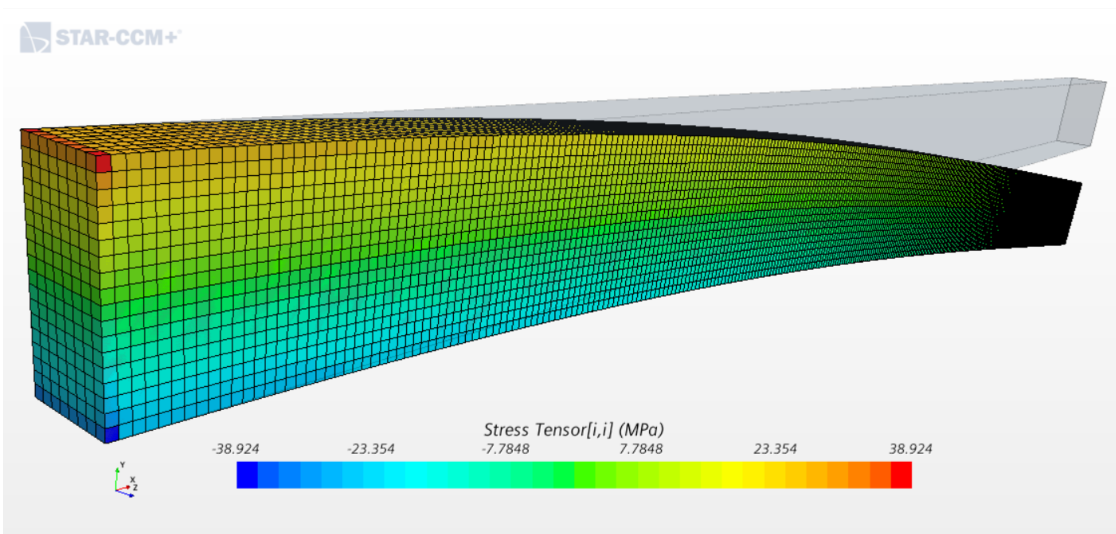
Figure 8: Maximum Bending Stress Report Results for Different Input Parts

The boundary bending stresses are always larger since they are farther from the neutral axis of the beam. Comparing the line element and fully-fixed boundary stresses to the beam theory result gives relative errors of 1.04% and 31.63%, respectively.

The large relative error for the fully-fixed BC is due to stress concentrations caused by restraining the in-plane displacements. Fig. 9b shows the stress concentrations occur in the corners of the cross-section at the fixed end of the beam. Since the corners want to displace in both the y- and z-axis directions more than any other points in the cross-section, the concentrations occur there. The line element BC avoids these problems by only restraining the y and z neutral axes of the cross-section, which won't displace due to bending.



(a) Line Element BC



(b) Fully-Fixed BC

Figure 9: Bending Stress σ_{xx} Shown on Deflected Beam

5 Conclusion

Analytical line element theories do not take into account any y-z plane displacements; however, in reality there will be displacements in the cross-section plane due to Poisson's ratio causing lateral strains, which act perpendicular to the bending strain. By allowing these strains to occur freely without restraint via utilizing the line element BCs, no additional stresses are induced, which allows the simulation to match analytical results.