Practical Guide

Structural Boundary Conditions using OnScale

Developed and curated by the Ansys Academic Development Team

Madhumita Saravana Kumar

education@ansys.com

Summary

Defining the right type of boundary conditions is a crucial part of setting up any simulation. This practical guide lists some important types of structural boundary conditions that can be used while setting up a simulation using OnScale. With the help of examples, the importance of defining both supports and loads while setting up structural simulations have been showcased.

This educational resource aims to teach students some important structural boundary conditions that are available in OnScale and how to choose the right ones while setting up a simple structural simulation.

OnScale Solve is a cloud simulation platform that combines powerful finite element solvers with scalable HPC cloud computing resources. It has an efficient and intuitive simulation workflow user experience with automatic meshing features that allows the user to focus on the simulation at hand.

Table of Contents

1. Introduction

For any finite element simulation, boundary conditions are essential for the simulation setup without which the problem cannot be solved. Boundary conditions represent how a structure interacts with its environment through constraints like a load or fixed point with no ability to move. They play a crucial role in producing accurate results and hence it is important to model them as close to reality as possible.

This practical guide will explain some important structural boundary conditions that are available in OnScale and when to apply them.

To aid in navigation of this document, this document has a full hyperlinked Table of Contents and specific embedded hyperlink icons shown here:

Table of Contents >>

Top of Section >>

Top of Table >>

2. Structural boundary conditions in OnScale

In the case of structural simulations there are two main types of boundary conditions [1]:

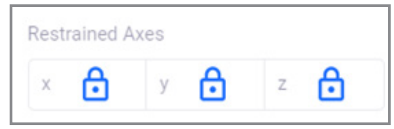
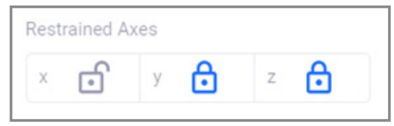
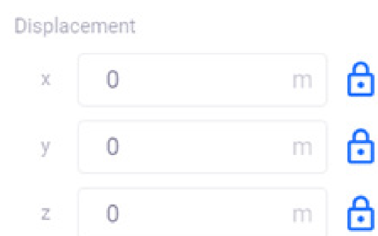
- Supports/Constraints – They are used to constrain the structure and limit the degrees of freedom in order to prevent rigid body motion.
- Loads – They are used to excite the structure which leads to deformation. These can be applied in the form of force, pressure, torques, etc.

OnScale offers a wide range of structural boundary conditions that can be used to setup simulation models. Some important ones have been described in the following sub-sections [2].

2.1 Supports or Constraints:

Here are four important types of constraints available in OnScale:

Table 1: Type of Supports/Constraints

Type of Constraint	Description	Available Options
	This type of support is used to limit the degrees of freedom both in terms of rotation and displacement.	*(1) Fixture:* Constraints all degrees of freedom about all 3 axes  *(2) Slider:* Allows displacement in only one axis  *(3) Restraint:* Allows displacement about the selected plane 
	This type of constraint is used to fix or prescribe the allowed displacement along an axis for a part or face. *Defaults units: m (meter)*	
	This type of constraint is used to fix or prescribe the allowed rotation about an axis for a part or face. *Defaults units: ° (degrees)*	
	This type of constraint applies a symmetry condition on a single or multiple co-planar surfaces. It is typically used to reduce the computational effort of the model where it is known that solution will be symmetrical.	

2.2 Loads

Here are four important types of loads available in OnScale [3]:

Table 2: Types of Loads

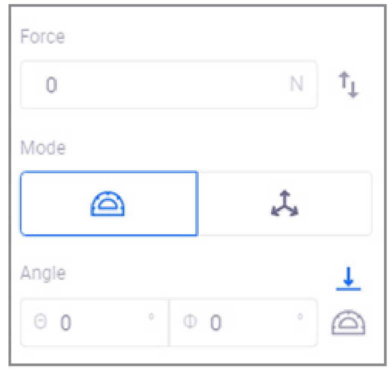
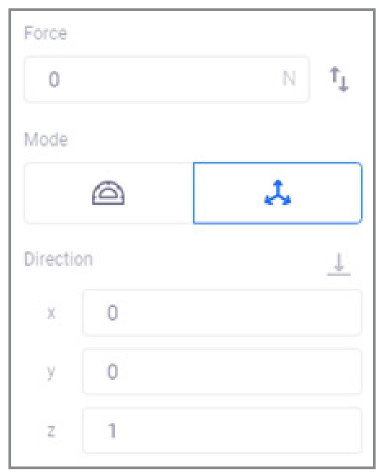
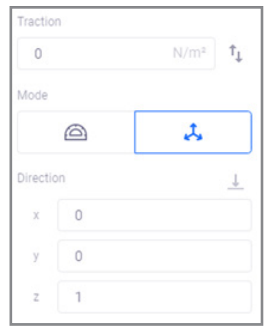
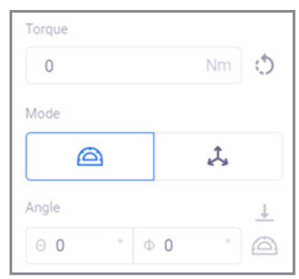
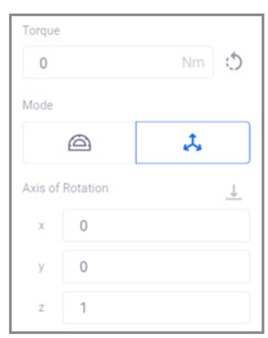
Type of Load	Description	Available Options
	This type of load is used to apply a force in a specified direction by defining the magnitude of the force and angle of application or the components of the direction vector of the force. *Defaults units: N (newton)*	*(1) Angle*: Force magnitude and angles can be defined  *(2) Direction:* Force magnitude and components of the direction vector can be defined. 
	This type of load is used to apply pressure perpendicular to a surface or multiple surfaces. *Defaults units: N/m² (Pascal)*	

Table of Contents >> Top of Section >>

Type of Load	Description	Available Options
	This type of load is used to apply a traction load (i.e. pressure + shear load) in a specified direction. It can be defined in terms of magnitude and angle or direction of the traction load. *Defaults units: N/m² (Pascal)*	*(1) Angle*: Traction magnitude and angles can be defined  *(2) Direction:* Traction magnitude and components of the direction vector can be defined 
	This type of load applies a torque/moment to a body. While defining a torque it is important to define the magnitude, direction of rotation (clockwise or anticlockwise) and axis of rotation. *Defaults units: Nm (Newton-meter)*	*(1) Angle:* Torque magnitude, direction of rotation and angles of axis of rotation can be defined  *(2) Direction:* Torque magnitude, direction of rotation and direction vector of axis of rotation can be defined. 

3. Examples of Boundary Conditions in Simulation Models

Here are a few examples of how the loads and constraints mentioned in the previous section can be used to simulate structural setups. To learn more about the whole simulation setup and meshing process refer to this lecture presentation Lecture Unit: Introduction to Meshing using Ansys OnScale.

The CAD models in the following examples are already available in OnScale CAD library.

3.1 I-Beam

Problem Statement

Consider a cantilever I-Beam with a load of 200N acting downward on its free end. Which boundary conditions need to be used to represent this problem?

Solution

A cantilever beam is a beam which is fixed on one end and free on the other. Therefore, we apply a support on one end where all degrees of freedom are constrained and a force load of 200N on its free end.

Step 1: Select the restraint tool and apply a fixture type of restraint on one end (see Figure 1).

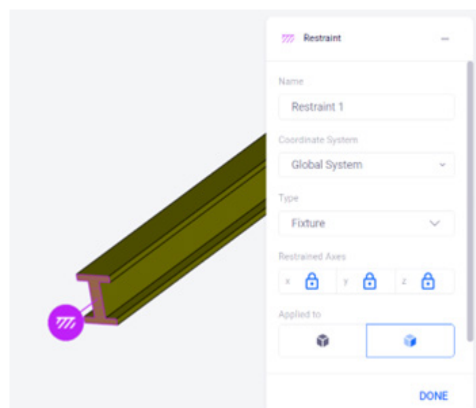


Figure 1: Using the restraint tool at one end of the beam

Step 2: Select the force tool and apply a load of 200N to the end face and provide the desired direction vector (see Figure 2).

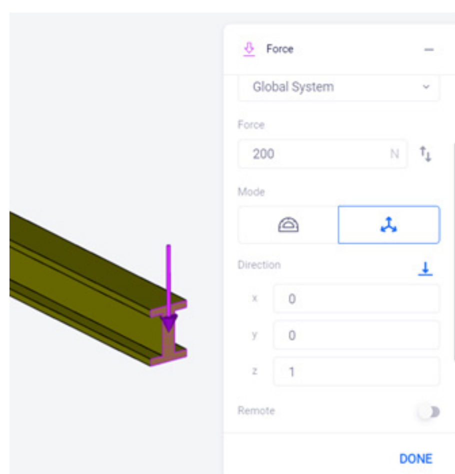


Figure 2: Using the force tool on the other end of the beam

3.2 Propeller

Problem Statement

Consider the propeller shown in Figure 3 fixed at the center. Apply a torque load of 1000Nm on one of its blades about the axis of the propeller.

Figure 3: Propeller Geometry

Solution

The propeller needs to be fixed at the center to prevent rigid body motion. A torque load of 1000Nm should be applied on one of its blades and the axis of rotation needs to coincide with the axis of the propeller.

Step 1: Select the restraint tool and apply a fixture type of restraint at the inner hub as shown in Figure 4.

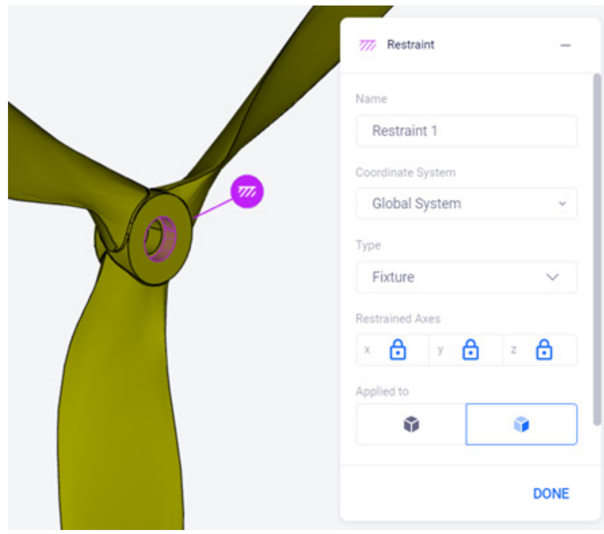


Figure 4: Using restraint tool at the inner hub

Step 2: Select the torque tool and apply a load of 2000Nm on one of its blades. Provide the desired direction vector as shown in the Figure 5.

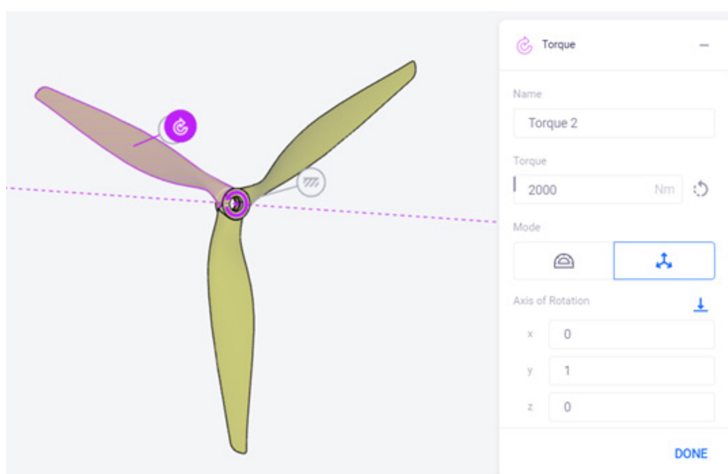


Figure 5: Using the Torque tool on one of the blades

3.3 Hollow sphere

Problem Statement

Consider a hollow sphere with an internal pressure of 1000 Pa. What boundary conditions need to be used to simulate only a part of the sphere in order to reduce computational effort (see figure 6).

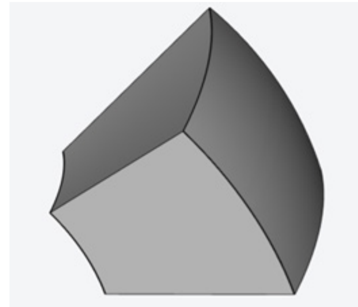


Figure 6: Partial hollow sphere geometry

Solution

To reduce computational effort, we simulate only a part of the hollow sphere. To facilitate this, symmetry boundary conditions are defined on the side faces and a pressure load of 1000 Pa is applied to the inner face of the hollow sphere.

Step 1: Select the symmetry tool and apply it to the four side faces of partial sphere as shown in figure 7.

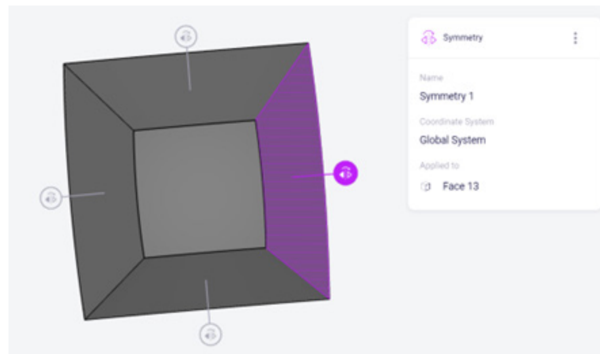


Figure 7: Using the Symmetry tool on the side of the partial hollow sphere

Step 2: Select the pressure tool and apply a load of 1000Pa to the inner face as shown in figure 8.

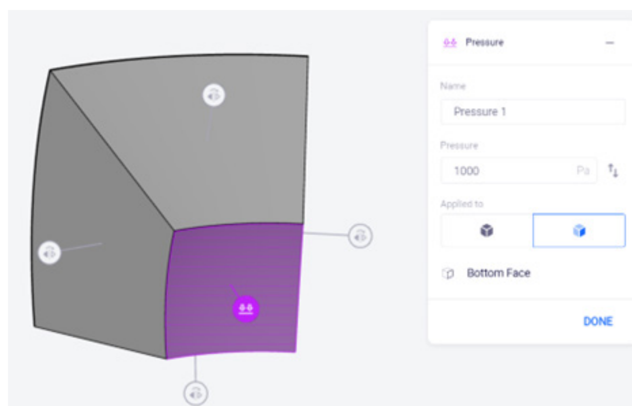


Figure 8: Using the Pressure tool on the inner surface of the sphere

4. Conclusion and Additional Information

This practical guide can be used as instructional material to understand how to apply the right type of boundary conditions while setting up simple structural simulation problems in OnScale. In the examples shown in the previous section we can observe that a combination of loads and supports are required to accurately simulate any physical process.

It is important to note that in addition to the boundary conditions mentioned in this guide, OnScale allows more complex constraints to be defined by coding them using the OnScale API. For more information please refer to Boundary Conditions & Loading – OnScale.

References

[1] M. Saravana Kumar, J. Plocher, Ansys Education Resource – 'Lecture Unit: Structural Analysis of Beams with Ansys Discovery', Lecture Unit: Structural Analysis of Beams with Ansys Discovery | Ansys

[2] Onscale Theory - Let's talk about boundary conditions | OnScale

[3] Onscale Solve Help - Properties panel | OnScale Solve Help

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

Document Information

This case study is part of a set of teaching resources to help introduce students to structures, fluids, or heat transfer (physics areas supported by Ansys OnScale).

Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

If you notice any errors in this resource or need to get in contact with the authors, please email us at education@ansys.com.

ANSYS, Inc.
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com