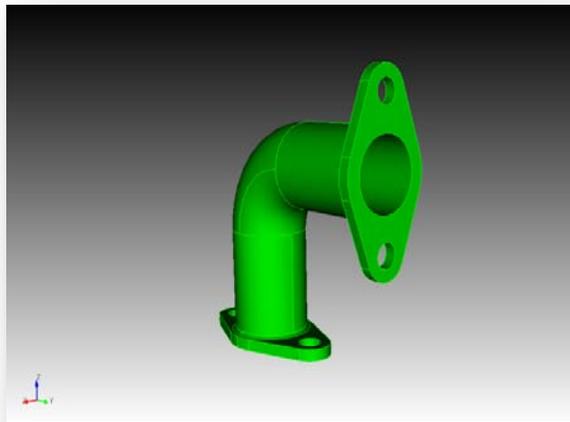# Trelis Boundary Layers Tutorial

Overview

In this tutorial, you will learn how to use the Trelis boundary layer feature to create a more specific mesh. A boundary layer is a user-created mesh with specific dimensions, growth rate, and number of layers. In this tutorial you will learn how to:

- Add a boundary layer to a simple geometric model
- How to use boundary layers with exterior flow
- How to use boundary layers with interior flow

Adding a boundary layer to a simple geometry.

Before creating a boundary layer, you need to create a geometry. For this example we will create a simple brick.
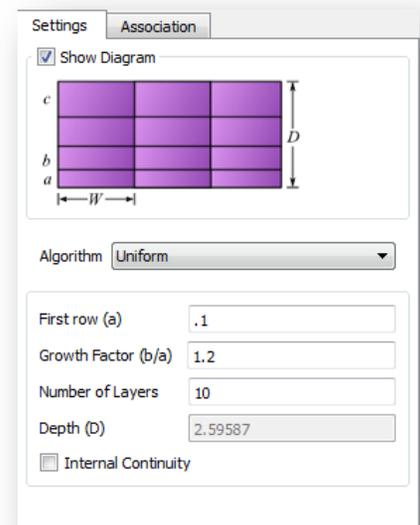
To create a brick

1. On the Command Panel, click on **Geometry** and then **Volume**.
2. Click on the **Create** action button.
3. Select **Brick** from the drop-down menu.
4. Enter **10** in the **X(width)** field. (10 should be there by default)
5. Click **Apply**.

Now that you have created a brick, you can add a boundary layer to it.

To create a boundary layer

1. On the Command Panel, click on **Mesh** and then **Boundary Layers**.
2. Click on the **Create** action button.

3. Click on the **Settings** tab.
4. Enter **0.1** in the **First row (a)** field.
5. Enter **1.2** in the **Growth Factor (b/a)** field.
6. Enter **10** in the **Number of Layers** field.

Note: First row is the size of the first row of the boundary layer. Growth Factor is the amount that it will grow by each layer. Number of Layers is the amount of layers in the boundary layer.
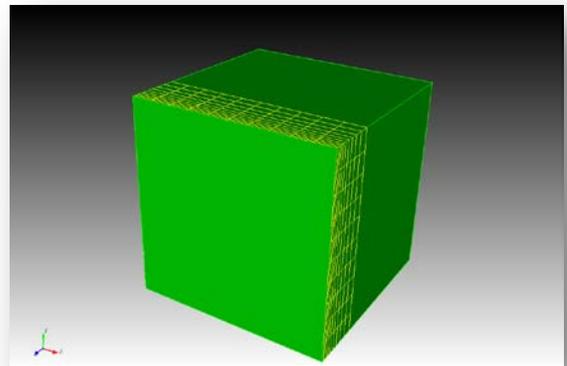
Now we have the specific measurements we want. We can apply these boundary layers to the specific surface we want.

7. Click on the **Association** tab.
8. Click the **Surface** radio button.
9. Enter **1** in the **Surface ID(s)** field.
10. Enter **1** in the **Volume ID(s)** field.
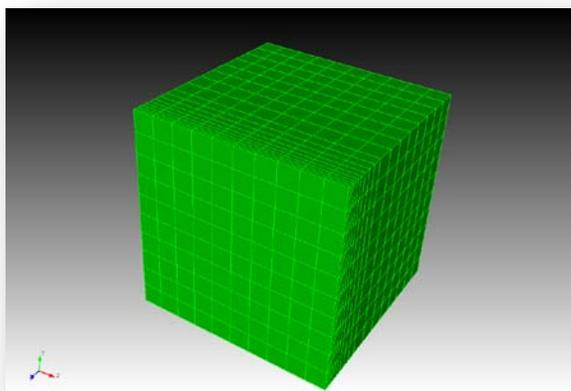11. Click **Add** and then **Create**.

With the boundary layer created, you can now mesh the brick.

To mesh the brick that has a boundary layer
1. On the Command Panel, click on **Mesh** and then **Volume**.
2. Click on the **Intervals** action button.
3. Enter **1** in the **Select Volumes** field.
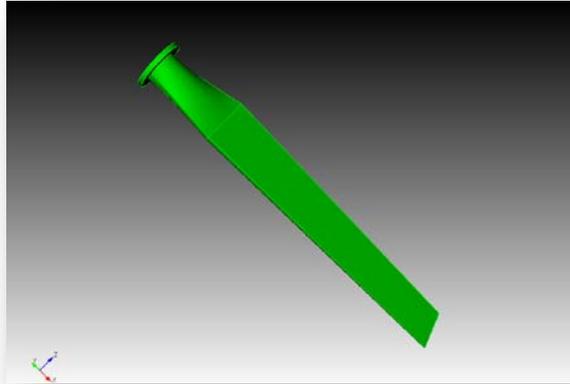4. Click **Apply** and then **Mesh**.

You will now be able to see the meshed model and the boundary layer in use. It should appear like the image shown below.

How to use a Boundary Layers For Exterior Flow Flow

In this example we will use a blade from a windmill. We want to analyze the exterior flow of air that would be making contact with the blade. We will use the shape of the blade to set boundary layers on the exterior part of the blade, which will represent the air.
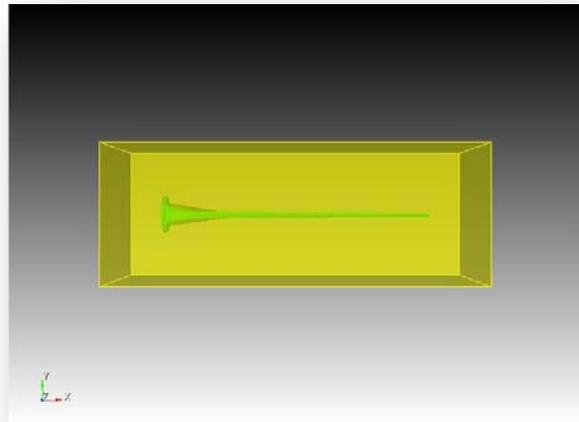

To import the model and set boundary layers

1. Click on the **File** and select **Import** from the drop-down menu.
2. Browse for the "Blade(clean).sat" file and click **Open**.

Note: You'll need to access this file from the Trelis web site under Support > Tutorials.

3. Leave the default options selected on the import dialog box and click **Finish**.
Note: It may be easier to accomplish the remaining steps with the view on **transparent** mode.
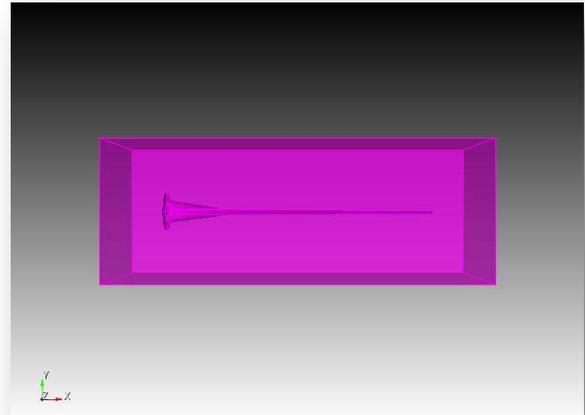
4. Click on the **View in transparent mode** button located on the toolbar.
5. On the Command Panel, go to **Geometry-Volume-Create** and select **Bounding Box** from the drop-down menu.
6. Enter **1** in the **Volume ID(s)** field. Choose E**xtended** and **Absolute**. Enter **150** for the value.

We have created a bounding box surrounding our model. This bounding box represents the air that the blade cuts through. To set boundary layers to the exterior part of the blade, we set the boundary layers on the air where it touches the blade.
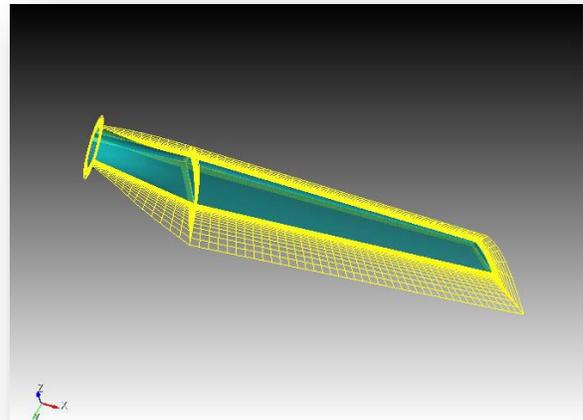
To delete the blade from the bounding box

1. On the Command Panel, click on **Geometry** and then **Volume**.
2. Click the **Boolean** action button.
3. Select **Subtract** from the drop down menu.
4. Enter **2** in the **A Body ID(s)** field.
5. Enter **1** in the **B Body ID(s)** field.
6. Click **Apply**.
7. Next, isolate the blade. click on **Geometry-Volume-Webcut** on the command panel.
8. Choose **Surface Extended** from the drop-down menu.
9. Enter **2** in the **Body ID(s)** field and **28** in the **With Surface ID(s)** field, then click **Apply**.
10. Delete volume **3**.

The remaining model is the air surrounding the blade. We can now set boundary layers the boundary layers to air surrounding the blade.
To set boundary layers to the exterior of the blade

1. Click on **Mesh** and then **Boundary Layer**.
2. Click the **Create** action button.
3. Click on the **Settings** tab.
4. Enter **0.05** in the **First row (a)** field.
5. Enter **1.2** in the **Growth Factor (b/a)** field.
6. Enter **10** in the **Number of Layers** field.
7. Tick the **Internal Continuity** box.
8. Click on the **Association** tab.
9. Click the **Surface** radio button.
10. Enter **20 to 26** in the **Surface ID(s)** field (all the interior surfaces).
11. Enter **4** in the **Volume ID** field.
12. Click on **Add** and then **Create**.

With the boundary layers created, set the size and mesh scheme.

1. In the Control Panel, click on **Mesh-Surface-Intervals**.
2. Enter **20 to 26** in the **Select Surfaces** field.
3. Select **Approximate Size** from the drop-down menu.
4. Enter **7** in the **Approximate Size** field and press **Apply**.
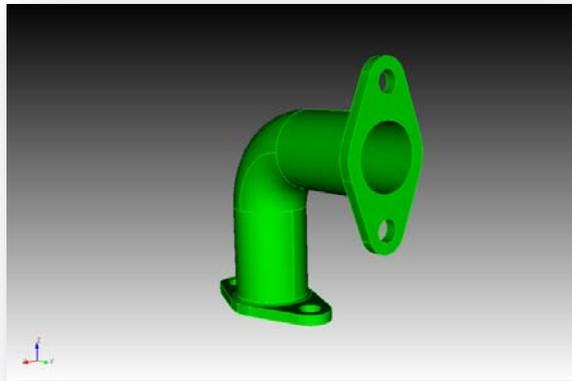5. In the Control Panel, click on **Mesh-Volume-Mesh.**

6. With the cursor in the **Select Volumes** field, click on the volume in the Graphics Window. The number **4** should appear in the field.
7. Select **Tetmesh** from the drop-down menu.x
8. Enter **1.0** in the **Growth Factor** field.
9. Press **Apply Scheme** and the, **Mesh**.

With the cursor in the Graphics Window, press the 'x', 'y', or 'z' key to slice through the mesh along an axis. Press the 'k' or 'j' key to move through the mesh layers. Press 'q' to exit this view.

As you can see from these steps, we have successfully set boundary layers to the exterior flow of the blade. By doing this we can effectively measure how the air moves around the shape of the blade.

How to use a Boundary Layers For Interior Fluid Flow

In this example we will use boundary layers to analyze interior flow. We will use a elbow pipe and set boundary layers to what would be the fluid flow inside the pipe.
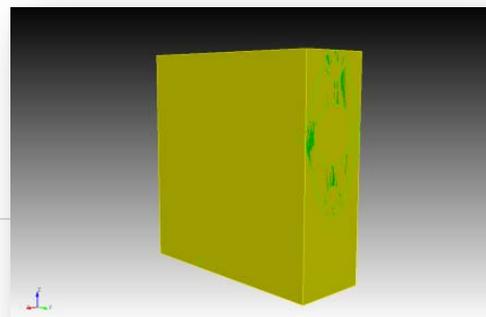

To import the model and set boundary layers

1. Click on the **File** and select **Import** from the drop-down menu.
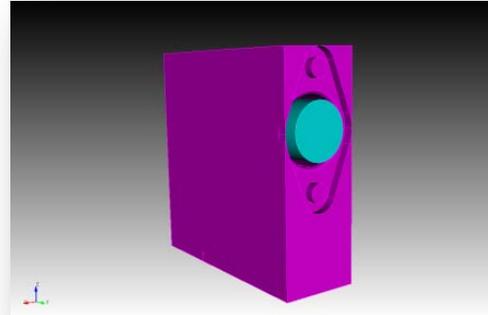2. Browse for the "**Bend.sat**" file and click **Open**.

Note: You'll need to access this file from the Trelis web site under Support > Tutorials.

Now that you can see the pipe, we want to analyze the fluid flow inside the pipe by setting boundary layers. We will first create a volume going through the pipe. Then we will remove the pipe, leaving us with the flow from inside the pipe.

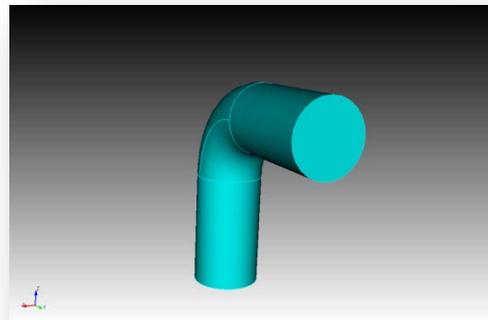To create a bounding box and remove the pipe

1. On the Command Panel, go to **Geometry-Volume-Create** and select **Bounding Box** from the drop-down menu.
2. Enter 1 in the Volume ID(s) field. Choose **Tight**.
3. On the Command Panel under **Geometry > Volume**, click on the **Boolean** action button.
4. Select **Subtract** from the drop-down menu.
5. Enter **2** in the **A Body ID(s)** field and **1** in the **B Body ID(s)** field, then press **Apply**.
6. In the command line, type **Split Body 2**.
7. Click the **Delete** action button.
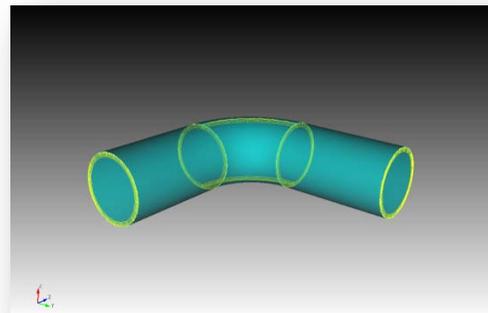8. Enter **3** in the **Volume ID(s)** field and click **Apply**.

The model that remains is the flow that exists inside the pipe. Now we will add boundary layers to the flow to help simulate the properties of the fluid inside.

1. Click on **Mesh** > **Boundary Layer** > **Create** action button.
2. Click the **Settings** tab.
3. Enter **.5** in the **First row (a)** field.
4. Enter **1.2** in the **Growth Factor (b/a)** field.
5. Enter **5** in the **Number of Layers** field.

Note: The Depth field number will auto-adjust based on the numbers entered in the previous fields. Be sure to adjust your numbers so the depth isn't greater than your model.

6. Click on the **Association** tab.
7. Click the **Surface** radio button.
8. Enter **123, 124, 125, 126, 127, 128,** and **129** in the **Surface ID(s)** field.
9. Enter **4** in the **Volume ID** field.
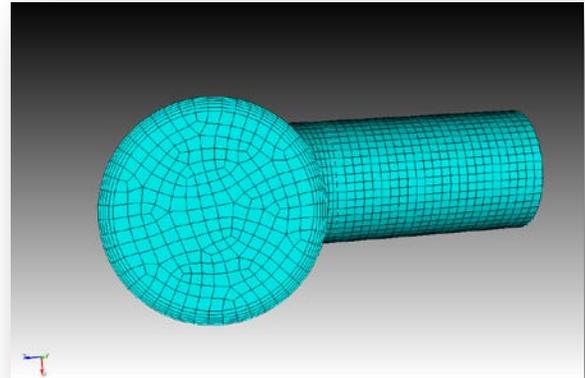10. Click **Add** and then **Create**.

Now that we have boundary layers throughout the model, we can mesh it.
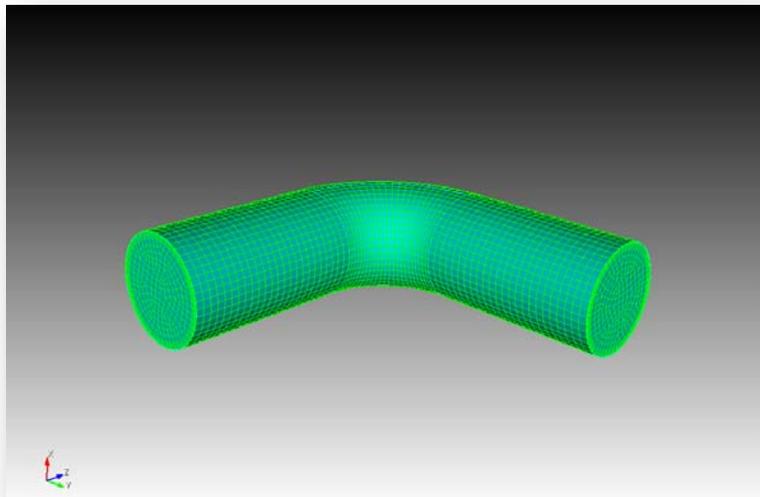To mesh the model

1. With the same settings on the Command Panel, click **Volume** and then the **Mesh** action button.
2. Enter **4** in the **Select Entities to Mesh** field.
3. Select **Sweep** from the drop-down menu.
4. Enter **121** in the **Source Surface ID(s)** field.
5. Enter **122** in the **Target Surface ID** field.
6. Click **Apply Scheme** and then click **Mesh**.

Note: The mesh size can be adjusted to generate a desired mesh.

7. Right mouse click on the mesh and click **Delete Mesh**.
8. Click the **Intervals** action button.
9. Enter **4** in the **Select Volumes** field.
10. Adjust the slider to specify a finer or coarser mesh.
11. Click **Apply** and then **Mesh**.

We have successfully set boundary layers and a mesh to the interior flow of the pipe.

Congratulations! You have finished the tutorial.