# Tutorial: Setting up a Linear Static simulation

Using the reciprocating saw design provided for this tutorial, you will set up a linear static simulation to evaluate the strength of the Blade Rod Connector.

## Open the Reciprocating Saw Design

1 If the Data Panel is not open, click Show Data Panel
.

2 In the Data Panel, open *1_ Setup_Linear_Static_simulation* from **Projects ➤ Samples ➤ Workshops & Events ➤ Adoption Path ➤ Test, Validate, and Explore Alternatives ➤ 1_Setup_Linear_Static_simulation.** The design appears on the Autodesk Fusion 360 canvas.
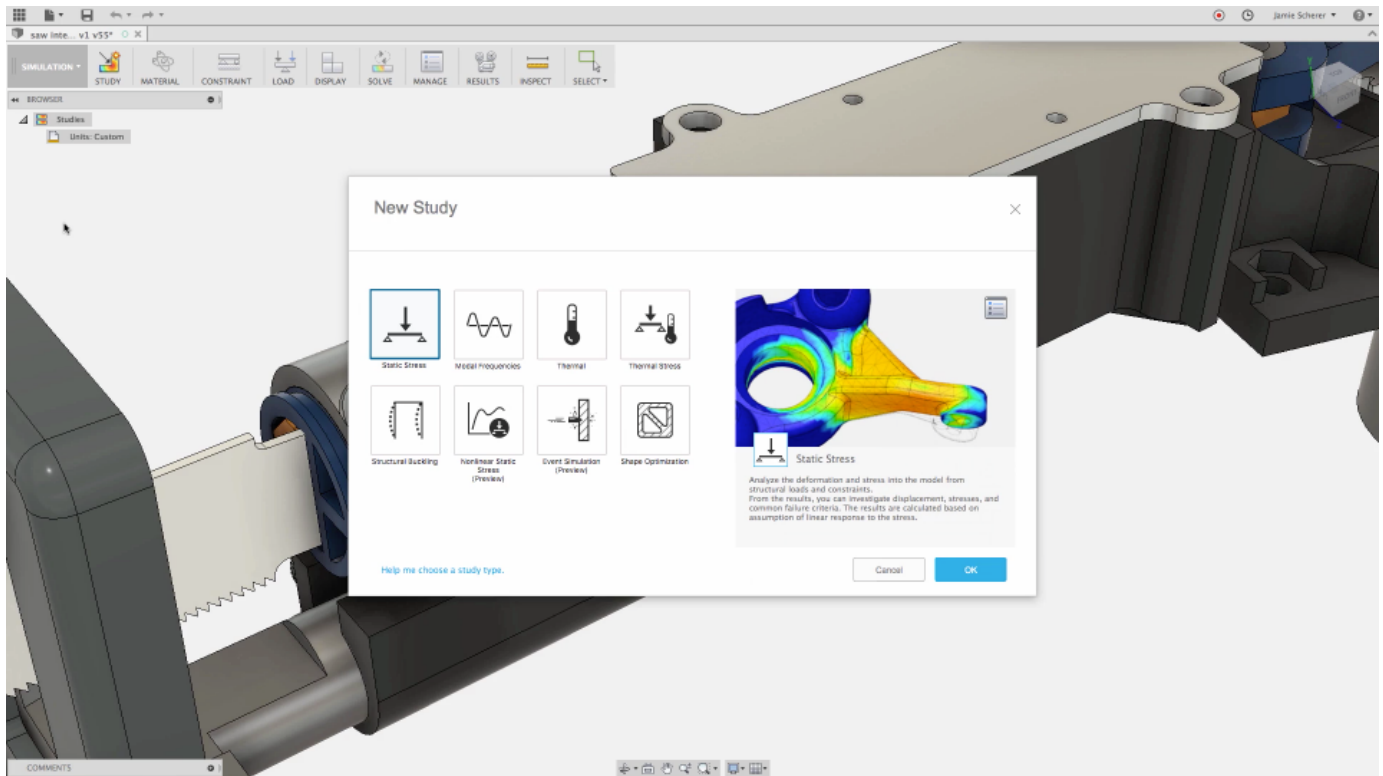
Switch to Simulation Workspace and Create New Study

Begin by switching to the simulation workspace in the workspace switcher. Select a Linear Static Study Type in the new study dialog, and Select the Settings icon and name the Study Under the General Settings to recall later.

1 In the Workspace Switcher, change to the Simulation Workspace.

2 Select Linear Static Study Type in the New Study
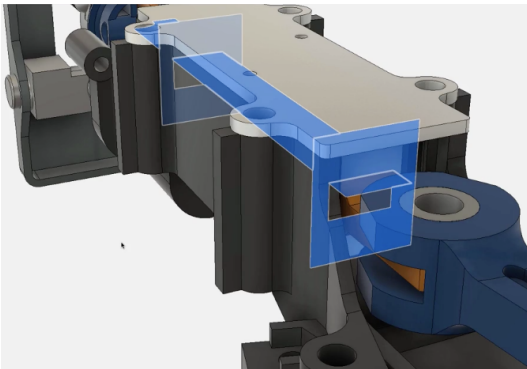Dialog.
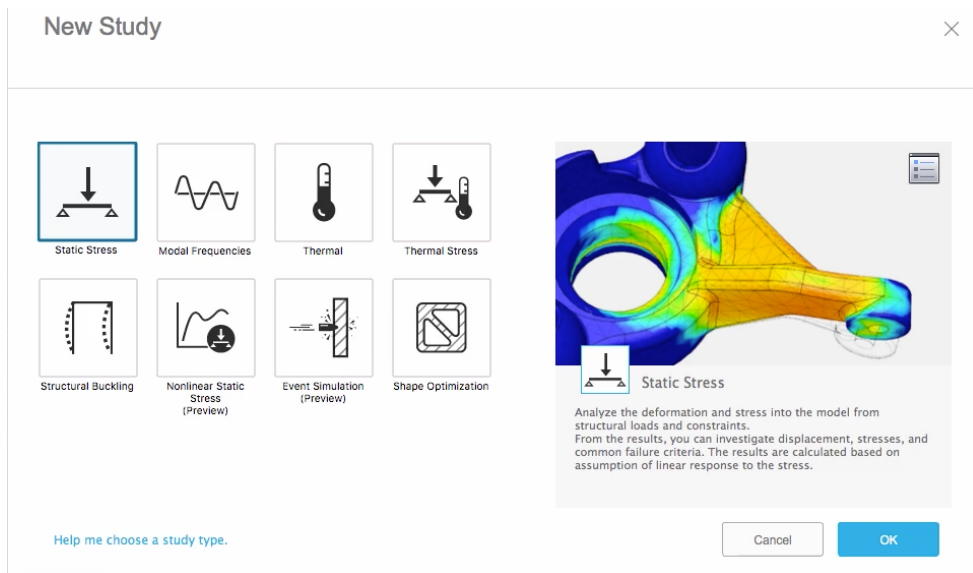
3 Select the **Settings** icon

4 Change the name of the study in the General Settings section.

5 Click OK to create Study Type.

A new simulation study has been added to the Browser.

Simplify Assembly for Simulation

Before setting up the boundary conditions for this simulation it is important to suppress all of the data we do not want included in the study.

1 In the Fusion window, Ctrl-click (Windows) or Command-click (macOS) the following:

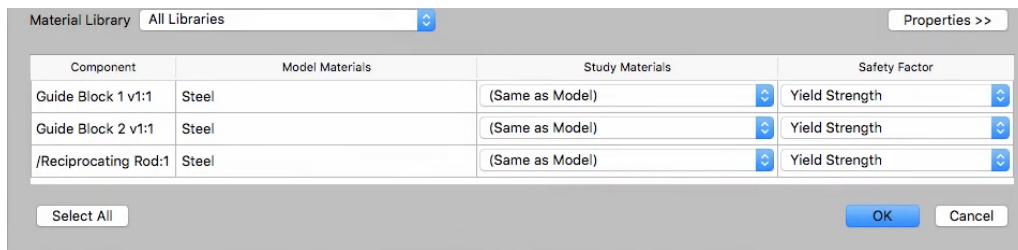- Blade Holder Assembly
- Guide Block 1
- Guide Block 2

2 Right-click the selections and choose Suppress All Except Selected.

# Assigning Study Materials & Structural Constraints

Begin by assigning the materials for all of the Components in your Study and then apply Fixed constraints to the hole in the Blade Rod Connector and the bottom of the Guide Blocks.

1 In the Toolbar select **Study Materials.**

2 Ensure that the Materials for all three components is set to Steel.

Material Library	All Libraries			Properties >>
Component	Model Materials	Study Materials	Safety Factor	
Guide Block 1 v1:1	Steel	(Same as Model)	Yield Strength	
Guide Block 2 v1:1	Steel	(Same as Model)	Yield Strength	
/Reciprocating Rod:1	Steel	(Same as Model)	Yield Strength	

Select All   OK   Cancel

3 Click OK

4 In the Toolbar open **Structural Constraints.**

5 With the Constraint Type Set to Fixed select the following:

- Hole in the Blade Holder Assembly
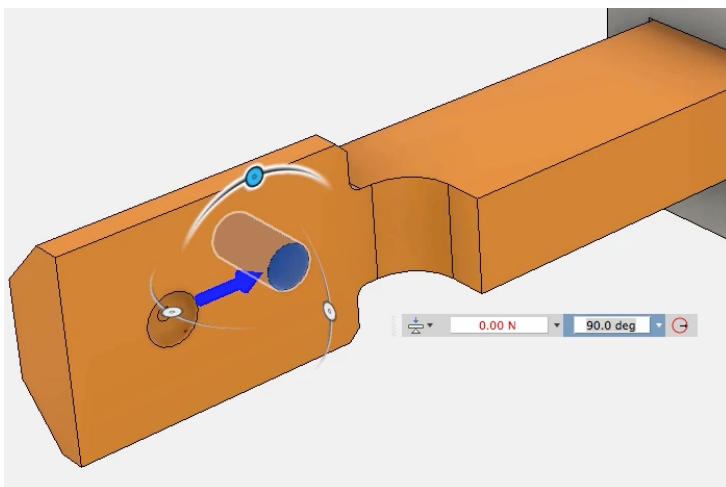- Bottom Face of Guide Block 1
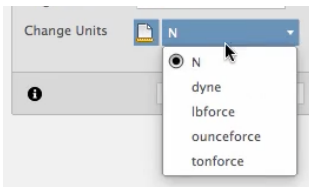- Bottom Face of Guide Block 2
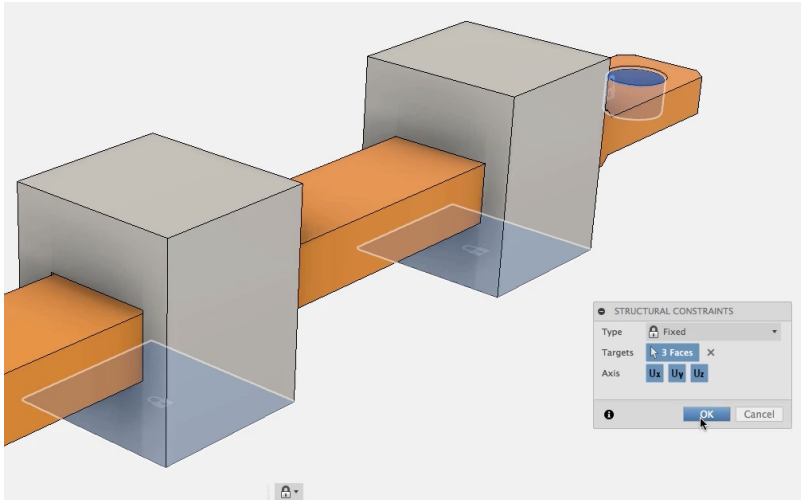
6 Click OK

Apply Structural Loads to the Study

Apply a 30lbforce compressive load on the Blade Rod Connector to simulate the load applied while in use.

1 In the Toolbar select **Structural Loads.**

2 With the Load Type set to Force, Select the hole that the Blade Assembly attaches to.

3 Change the X Angle to 90 deg.


4 Change the Units to lbforce.

   **5** Set the Magnitude to 30 lbforce.

    6 Click OK

Clone Study to Create More Studies to Solve

Clone the existing Study twice to create two other load cases we can solve. All existing boundary conditions will copy to the Cloned Studies and allow you to change their values.

    1 In the Browser Right-Click on the existing Study

    2 Select Clone Study

    3 *Rename Study as desired.*

    4 In the Browser, expand the **New Study ➤ Load Case ➤ Right-Click on Load and Select Edit Structural Load.**
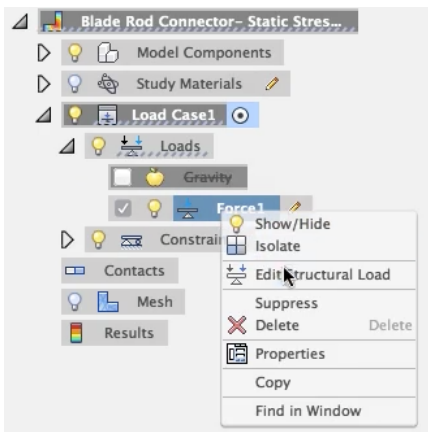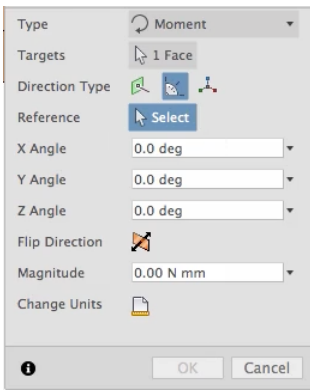

    5 Change the X Angle to 180 deg.
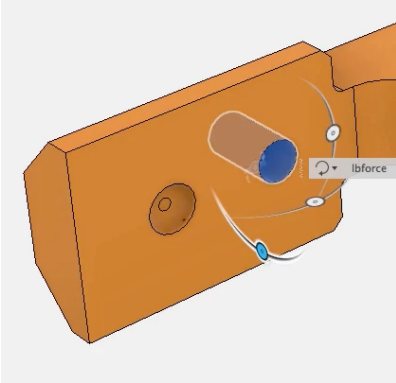
    6 Click OK.
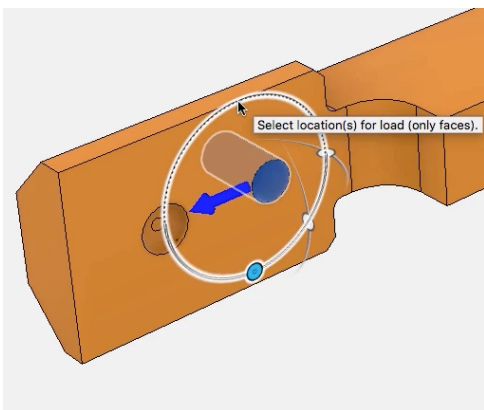
Next repeat the previous steps to Clone another Study for another Load Case.
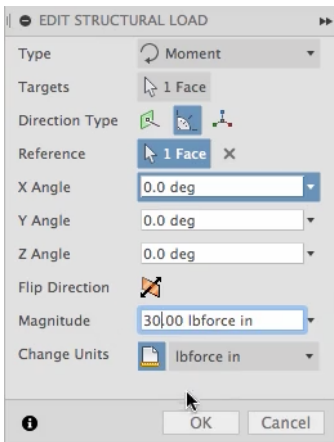
    1 In the Browser Right-Click on the existing Study.

    2 Select Clone Study.

    3 Rename Study as desired.

    4 In the Browser, expand the **New Study ➤ Load Case ➤ Right-Click on Load and Select Edit Structural Load.**

    5 Change Load Type to Moment.

Type	⟳ Moment ▾
Targets	▷ 1 Face
Direction Type	◻ ▨ ⊥
Reference	▸ Select
X Angle	0.0 deg ▾
Y Angle	0.0 deg ▾
Z Angle	0.0 deg ▾
Flip Direction	⊠
Magnitude	0.00 N mm ▾
Change Units	▯

| ❶ | OK | Cancel |

6 Reselect the Target as the hole that the Blade Assembly attaches to.

7 Set the Magnitude to 30 lbforce in.

● EDIT STRUCTURAL LOAD ▸▸

Type	⟳ Moment ▾
Targets	▷ 1 Face
Direction Type	◻ ▨ ⊥
Reference	▸ 1 Face ✕
X Angle	0.0 deg ▾
Y Angle	0.0 deg ▾
Z Angle	0.0 deg ▾
Flip Direction	⊠
Magnitude	30.00 lbforce in ▾
Change Units	▯ lbforce in ▾

| ❶ | OK | Cancel |

8 Click OK.

Generate Mesh

Generate the Finite Element Mesh in preparation to Solve the studies created.

1 In the Toolbar, navigate to **Solve ➤ Generate Mesh.**