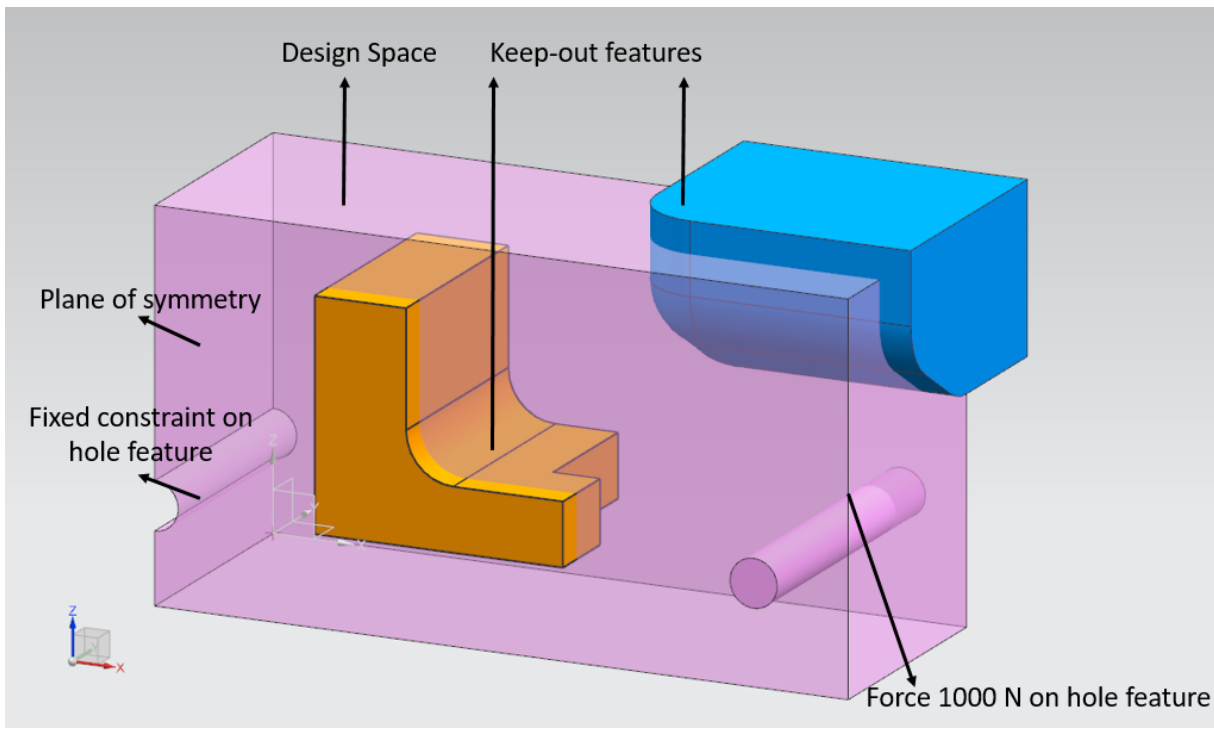# TOPOLOGY OPTIMIZATION

This tutorial shows how to perform **Topology Optimization** on the part shown, with specified loads and constraints

In **NX**, on the **Ribbon bar**, in the **File** menu, click on **Open**. Browse for **Simple_Cantilever.prt** in the folder in which it was downloaded. Select the part file

NOTE: To understand the relevant terminology in this tutorial associated with the Topology Optimization workflow, go to **Topology Opt → Topology Opt Help**
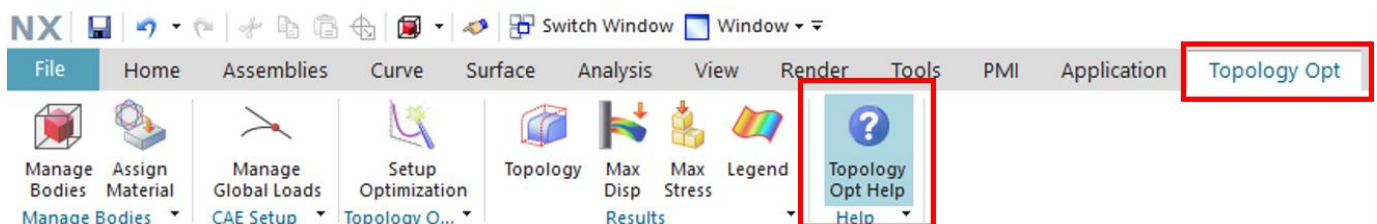
TIP:
- Each dialog box in the Top Opt workflow may retain dialog memory from previous iterations. If it is found that a certain parameter in a dialog box is

already set without the user having assigned a value, the dialog memory needs to be refreshed. Click on ⟳ icon in a dialog box to refresh memory

- If you need to reset the entire Topology Optimization workflow in the **Manage Bodies** dialog box, click on **Reset Topology Optimization workflow**
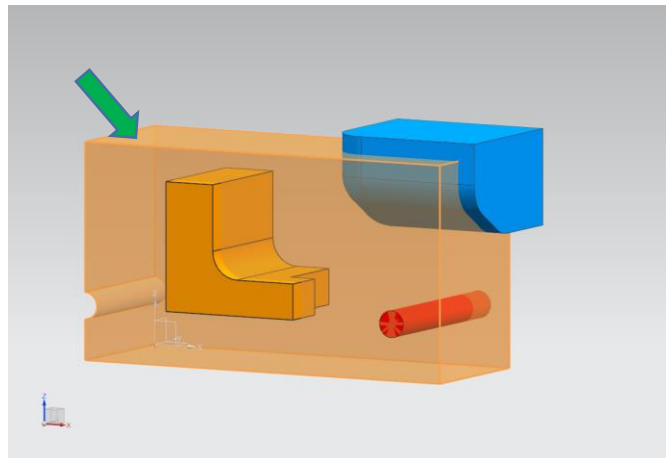
MANAGE BODIES

1. In the Modeling application, select **Topology Opt**
2. In the Ribbon bar, select **Manage Bodies**.

3. In the graphics window, select the Design Space body to be optimized
   NOTE: For more information on what is a 'Design Space' refer **Topology Opt Help | Manage Bodies | Bodies**

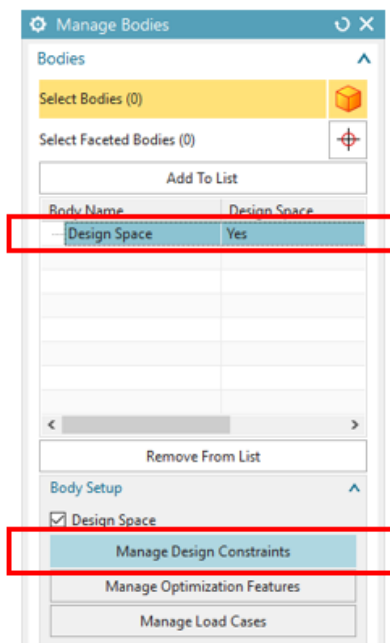   **Notice** the Design space highlighted in the graphics window

4. In **Bodies group,** click **Add To List**
   Notice the Design Space body has been added to the **Bodies list**
5. To assign a body from the list as design space, activate the **Design Space** checkbox
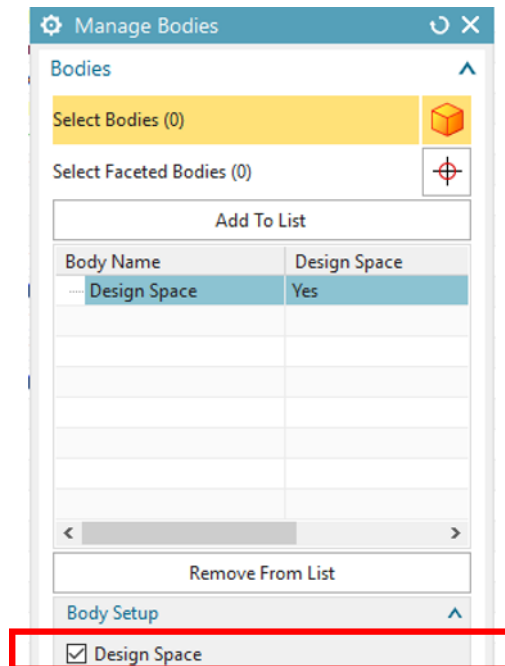
MANAGE DESIGN CONSTRAINTS

1. Select the design space from the list, click **Manage Design Constraints**

2. From the **Constraints Type** drop down list, choose **Planar Symmetry**
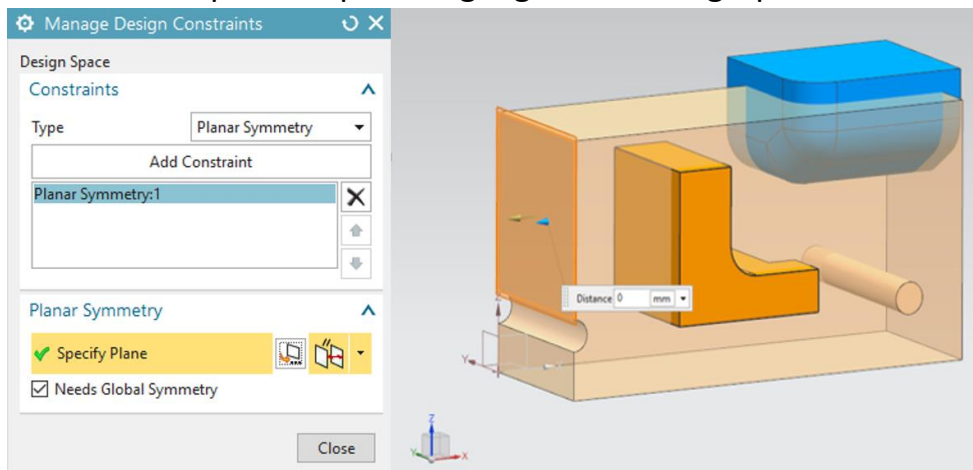
NOTE: For detailed information on '**Planar Symmetry Constraint**' go to **Topology Opt Help | Manage Bodies | Bodies | Manage Design Constraints | Planar Symmetry**

3. Click **Add Constraint**
   Notice the Planar Symmetry constraint is added to the Constraints list

4. Select the constraint from the list and define the parameters for the constraint. For **Planar Symmetry** constraint, define the plane about which symmetry is desired. **Needs Global Symmetry** checkbox is enabled
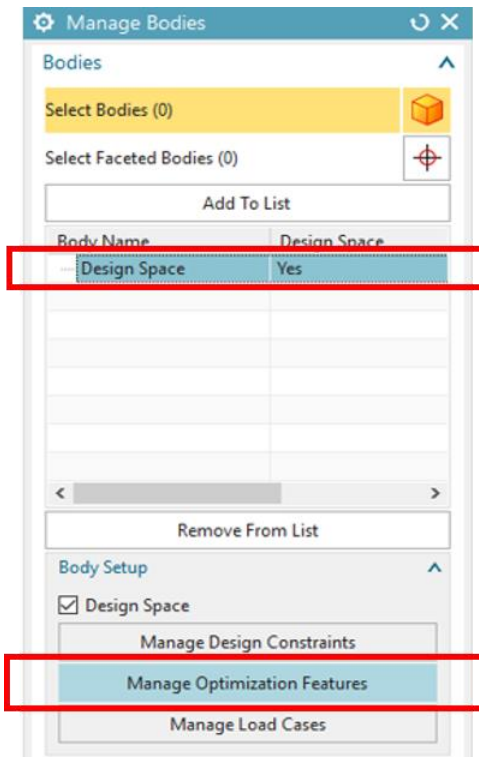
Notice the specified plane highlighted in the graphics window

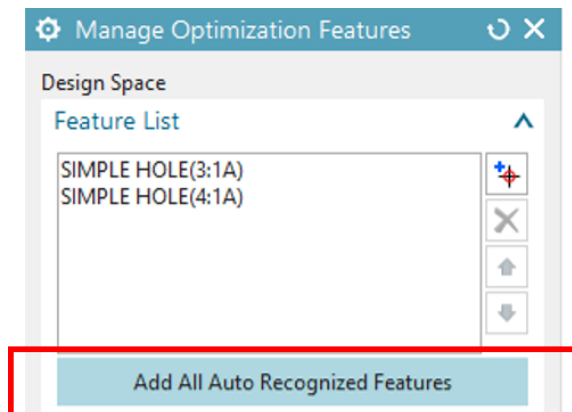5. Close the **Manage Design Constraints** dialog box

MANAGE OPTIMIZATION FEATURES

1. Select Design space from the list, click **Manage Optimization Features**

2. In **Manage Optimization Features** dialog box, in the **Feature List group,** click **Add all Auto Recognized Features.**
   **Notice** some features have been added to the **Feature list**

<u>NOTE:</u> For detailed information on '**Auto Recognized Features',** go to **Topology Opt Help | Manage Bodies | Bodies | Manage Optimization Features**

3. To add more feature bodies to the **Feature list**, click on ⊕ symbol, select solid bodies from the graphics window, and click OK.
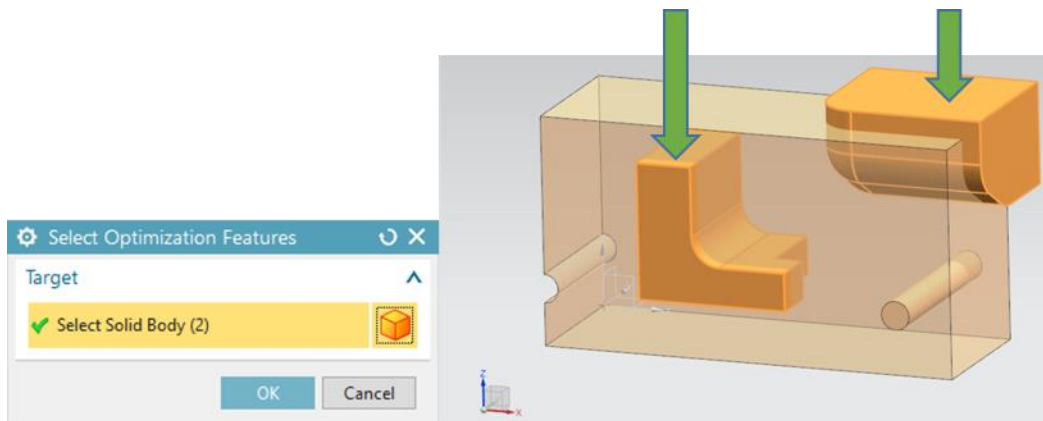
Notice the feature bodies highlighted in the graphics window
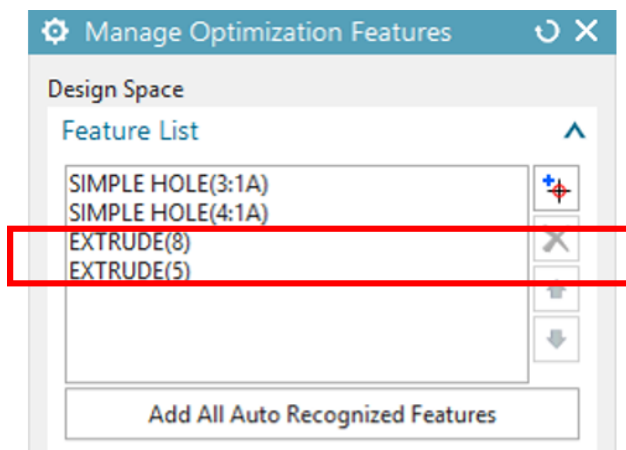
Notice the updated **Feature List**

4. From the **Feature list** select a feature
5. In **Feature Properties** group apply the parameters and settings for each feature as described in the chart below

This chart specifies the feature properties for each optimization feature

| Design Space	Feature List	Feature Properties	Maximum Displacement
Design Space	SIMPLE HOLE (3:1A)	Keep In/Out : Out Auto Blending Distance : Enabled Offset Thickness : 0.5 mm Type of Constraint : Fixed	Disabled
	SIMPLE HOLE (4:1A)	Keep In/Out : Out Auto Blending Distance : Enabled Offset Thickness : 0.5 mm Type of Constraint : No Constraint	Disabled
	EXTRUDE(8)	Keep In/Out : Out Auto Blending Distance : Enabled Offset Thickness : 0 mm Type of Constraint : No Constraint	Disabled
	EXTRUDE(5)	Keep In/Out : Out Auto Blending Distance : Enabled Offset Thickness : 0 mm Type of Constraint : No Constraint	Disabled

.

NOTE: For detailed information on the effect of each of the above parameters on the result, refer **Topology Opt Help | Manage Bodies | Bodies| Manage Optimization Features**

MANAGE LOAD CASES

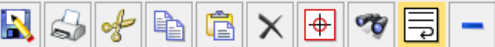1. Select Design space from the list, click **Manage Load Cases**
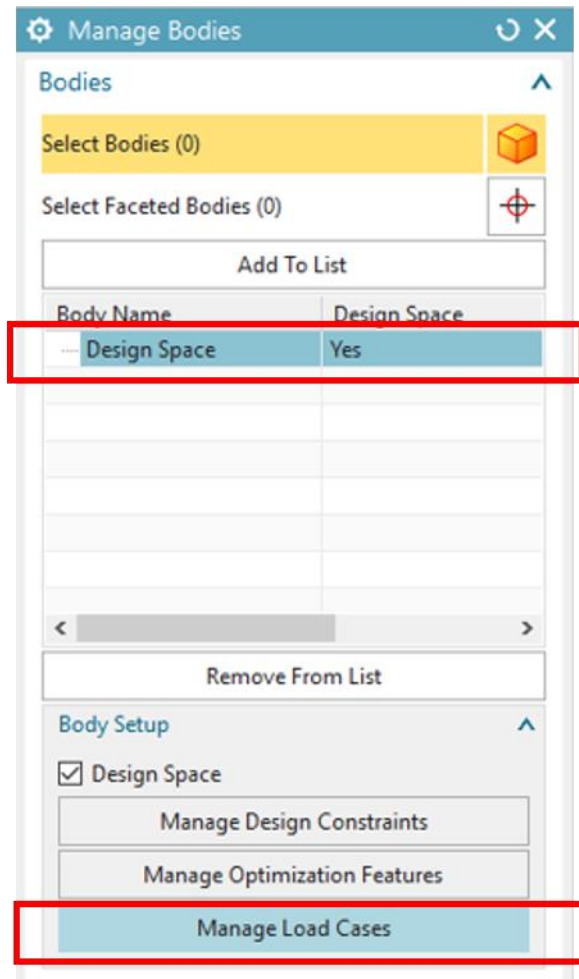
2. In the **Manage Load Cases** dialog box, in **Features group,** the list of features is auto populated
3. From the **Features** list select **SIMPLE HOLE(4:1A)** feature
4. In the **Loads group -**
- Select the load **Type** to be applied from the drop-down list. Select-
  Type = Force
- Specify the magnitude and direction of the load. Enter-
  Force = 1000 N
  Direction Vector = (0.0, 0.6, -0.8)
  NOTE: For detailed information on different types of loads, refer **Topology Opt Help | Manage Bodies | Bodies | Manage Load Cases | Load Type**
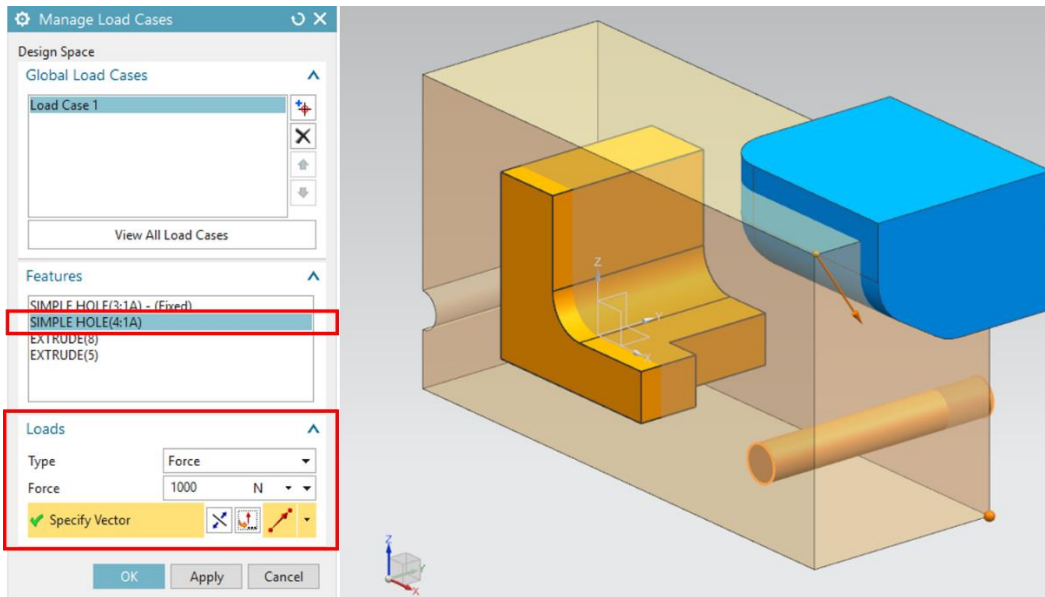
Notice in the graphics window, the direction of load on highlighted feature

NOTE: In the **Global Load Cases group,** click on **View All Load Cases** to view the type, magnitude and direction of loads applied on all features in all load cases. There is always a single load case (**Load Case 1**) by default
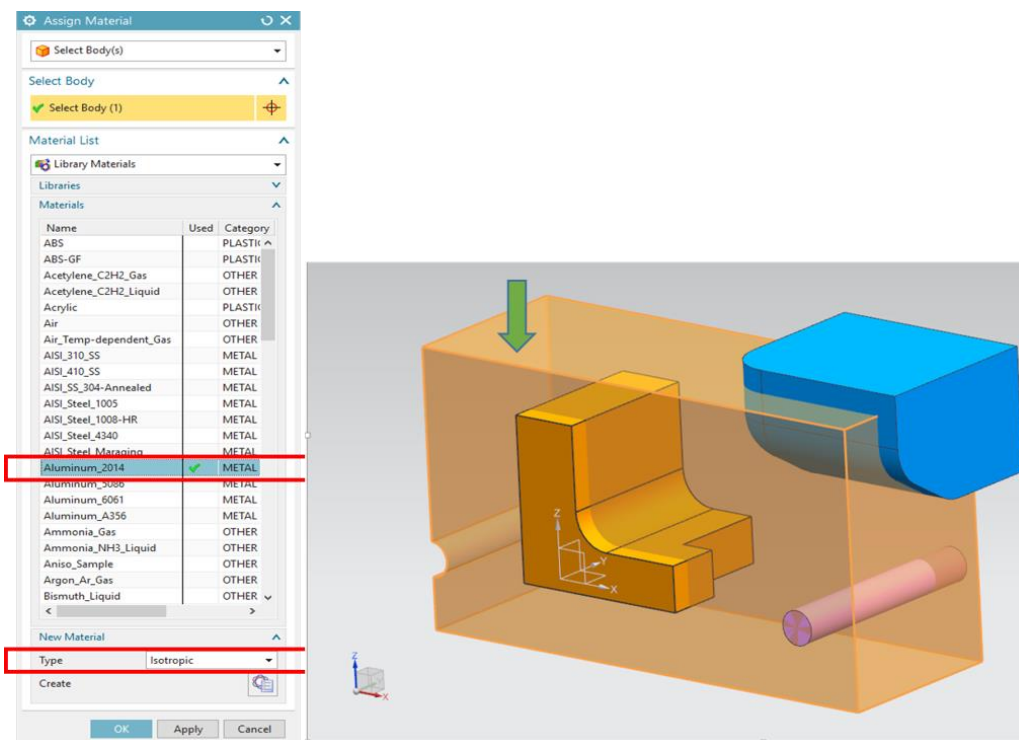
ASSIGN MATERIAL

1. Each body in the **Bodies** list needs to be assigned a material. In the Modeling application, select **Topology Opt**

2. In the Ribbon bar, select **Assign Material**
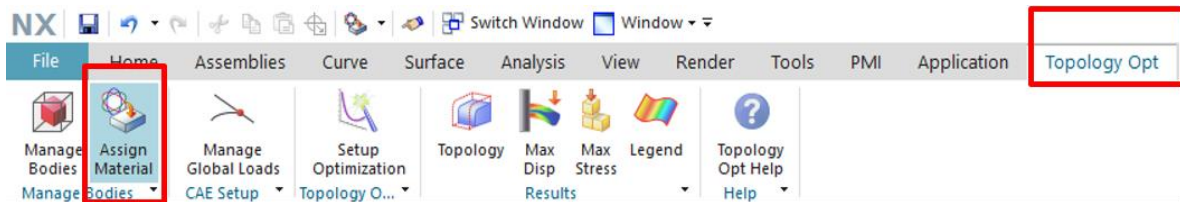
3. In the graphics window, select Design Space to assign a material to it

4. In the **Assign Material** dialog box, under **Material List** group**,** select **Aluminum_2014** from the **Materials** list

5. In the **New Material** group**,** from the drop-down list of material properties, select **Type = Isotropic**
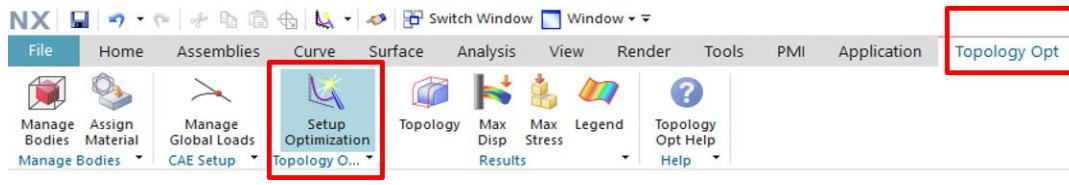

| Design Space	Assigned Material	Material Type
Design Space	Aluminum_2014	Isotropic

6. Click **Apply/OK**

SETUP OPTIMIZATION

1. In the Modeling application, select **Topology Optimization**
2. In the Ribbon bar, select **Setup Optimization**

3. In **Setup Optimization** dialog box, **Parameters** tab, in the **Optimization group,** select the optimization type from the drop-down list
    Optimization Type = Minimize Strain Energy subject to mass target
4. In **Select Global Resolution** group**,** move the slider to select a resolution towards **Fast/Coarse**
5. After the above two steps, click **Estimate Optimization Parameters.** The values under **Optimization Parameters** are determined
6. Select the body from the **Design Spaces** group**,** under **Optimization Constraints group,** enter a numeric value for **Mass Target**
    Mass Target = 0.02 kg
7. Click **Run Optimization**
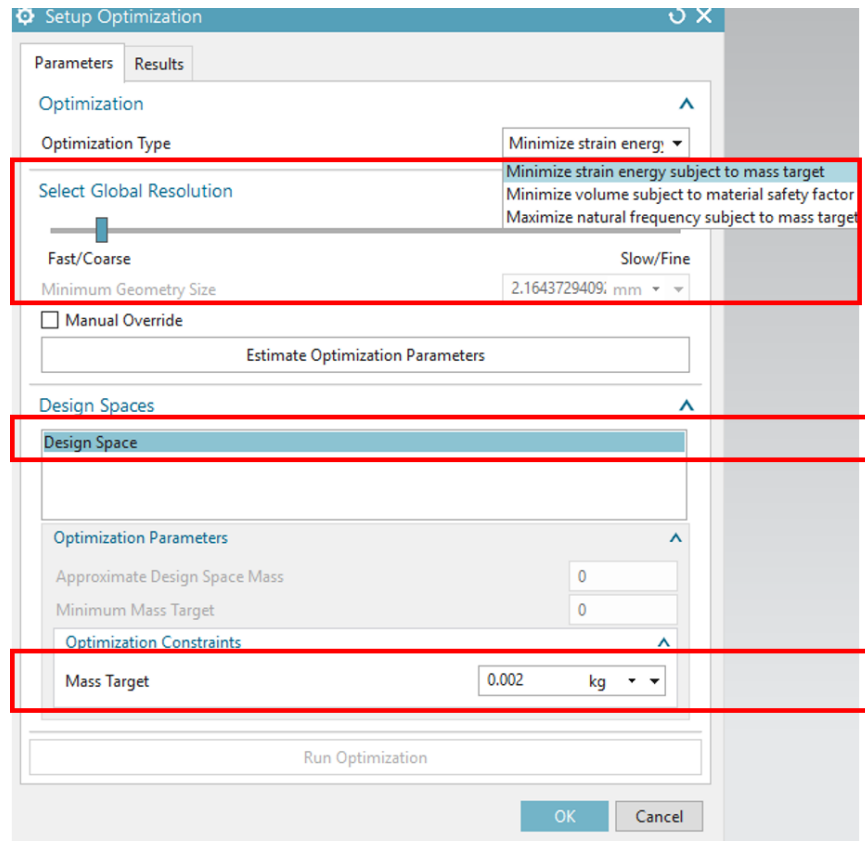
| Optimization Type	Select Global Resolution	Design Spaces	Optimization Constraints
Minimize strain energy subject to mass target	Fast/Coarse	Design Space	Mass Target : 0.002 kg

RESULTS

1. As the optimization runs, the **Results** tab displays-
2. **Diagram of Convergence :** This shows the progress of the optimization in two stages of convergence to the optimal design
3. **Log:** The log displays- **Errors** found in the optimization setup (if any), the **displacement** and **stress** results of the optimization, the time taken after the optimization is complete.
4. **Status :** The status shows whether the optimization is **Running, Meshing** or **Finished**
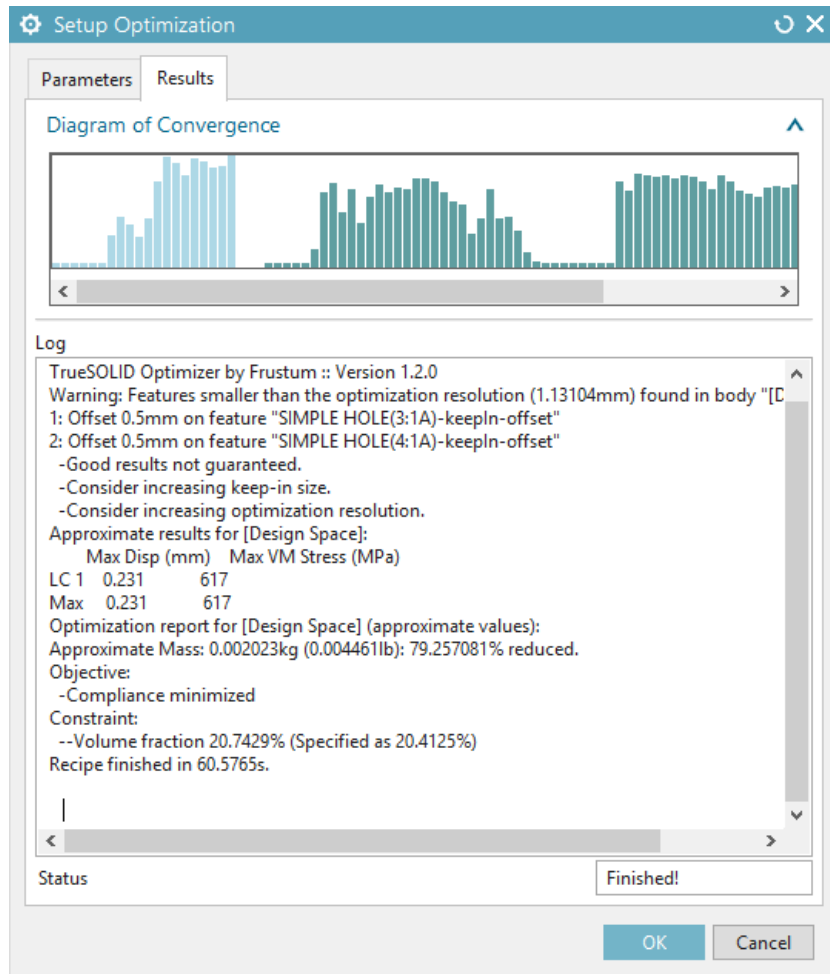
5. In **Topology Opt → Results group,** toggle through the **Topology, Max Disp, Max Stress** to view simulation results of the optimized geometry
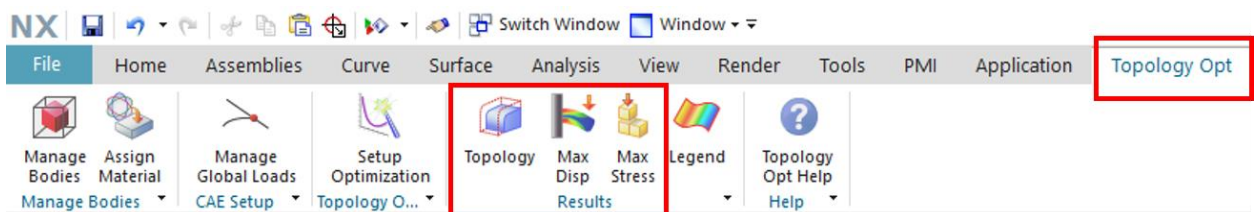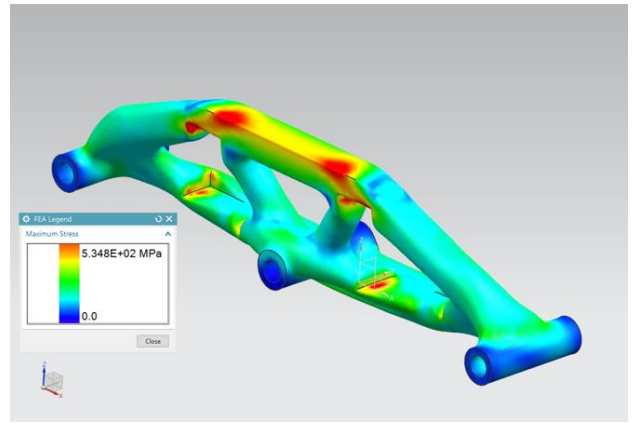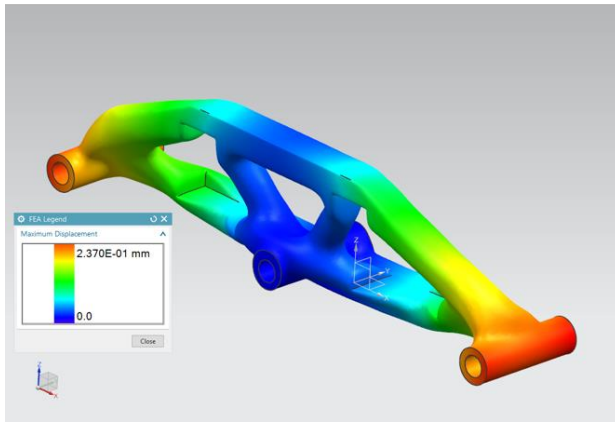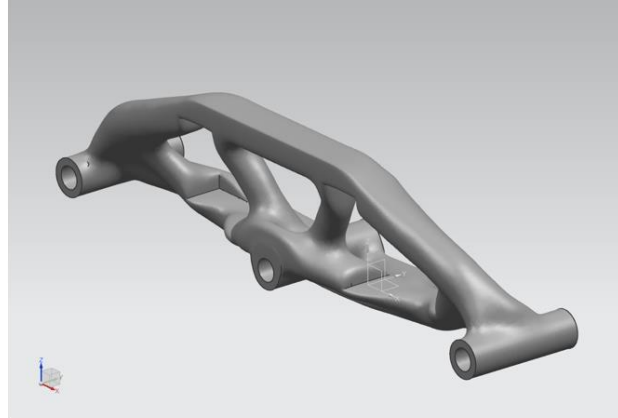
NOTE: For more information on **Setup Optimization** criteria and analysis of top opt results, refer **Topology Opt | Manage Bodies | Setup Optimization**