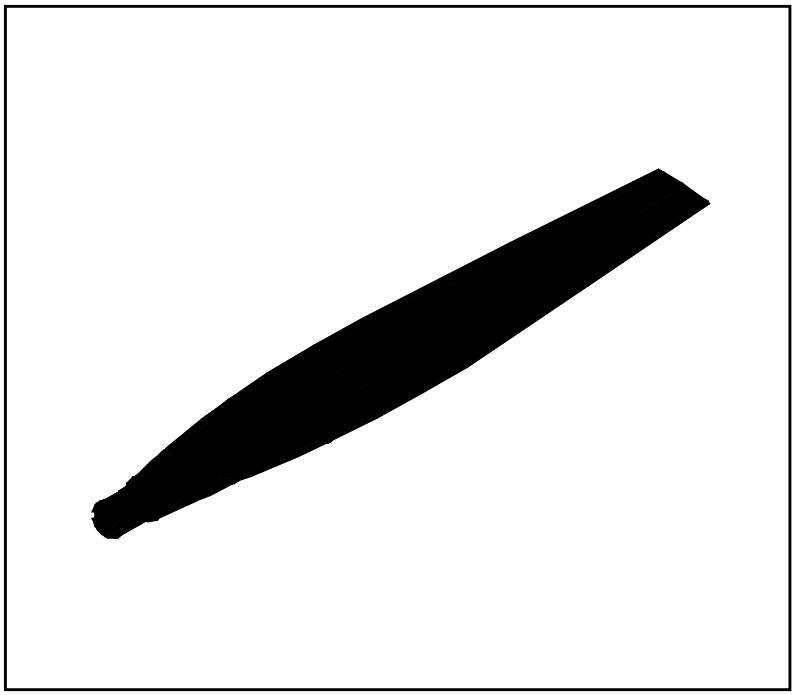
LESSON 11

Mass Properties Calculations



Objectives

- Import a unigraphics express file and apply mass properties to the propeller.

Suggested Exercise Steps

- Open a new database and name it fin.db.
- Import the Express Neutral file prop_11.1.exp
- Create the material properties for steel and apply it to the model
- Review the Mass Properties
- Create a projection of the model, then create a surface from that projection. Define a field to represent the thickness of the model
- Again review the Mass Properties and compare it to the previous results
- Mesh the base of the solid

Exercise Procedure

1. Create a new database named **fin.db**.

Open a New Database

File/New ...

New Database Name: **fin**

OK

The *New Model Preferences* form appears.

Tolerance: ◆ **Based on Model**

Analysis Code: **MSC/NASTRAN**

Analysis Type: **Structural**

OK

2. Import an Express Neutral file.

Import a Express Neutral File

File/Import...

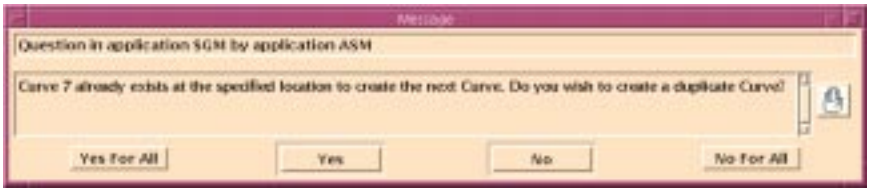
Object: **Model**

Source: **Express Neutral**

Import File: **prop_11.1.exp**

Apply

A message form appears.



No For All

Close the *Import Summary* form.

OK

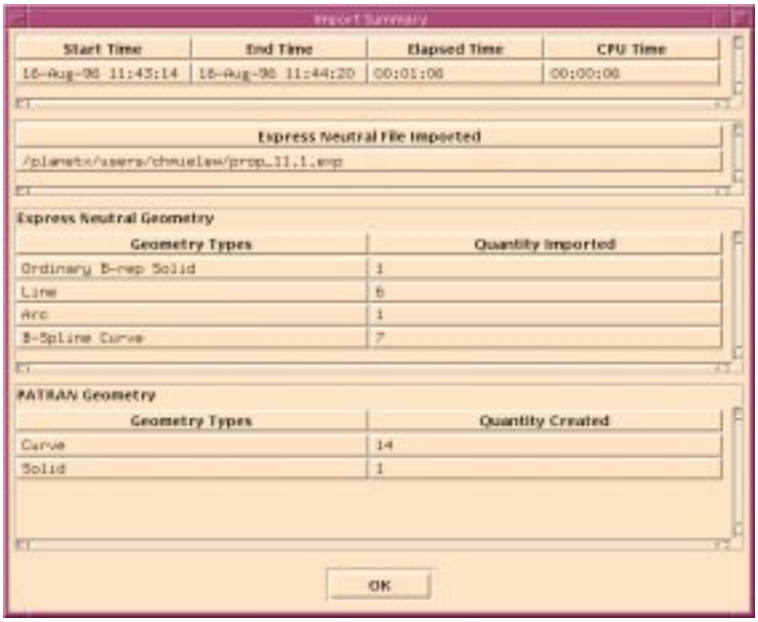
The *Import Tolerances* form appears. A tolerance mismatch may have caused the model to import improperly. The model should therefore be re-imported with a new model tolerance.

Apply

A message form appears. Again, do not duplicate geometry.

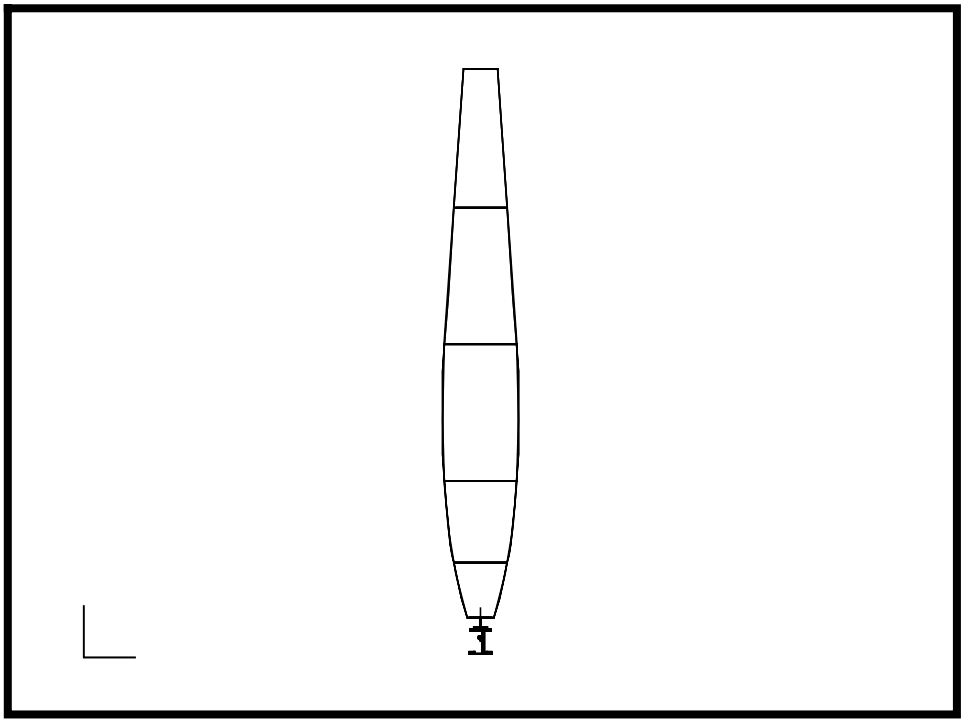
No for All

A new *Import Summary* form appears.



OK

The imported model should appear as shown below.



3. To keep the imported solid separate from geometry that will be created later in the exercise, create a group named **solid** and add all solids to it.

Group/Create...

Action: **Create**

New Group Name: **solid**

■ **Unpost All Other Groups**

In order to select **Solid 1**, use the following entity select icons:

Geometric Entity

Solid

Entity Selection: **Solid 1**

Apply

Cancel

Create a New Material

4. Create a material named **steel**.

◆ **Materials**

Action: **Create**

Object: **Isotropic**

Method: **Manual Input**

Material Name: **steel**

Input Properties...

The *Input Options* form appears.

Elastic Modulus: **30e6**

Poisson Ratio: **0.3**

Density: **0.0029**

Apply

The phrase **Linear Elastic - [,,,,] - [Active]** appears in the *Current Constitutive Models* box.

Cancel

5. Create a property set to apply to the solid model. Name it **fin_3d_solid**.

Create a Property Set

◆ **Properties**

Action:	**Create**
Dimension:	**3D**
Type:	**Solid**
Property Set Name:	**fin_3d_solid**

Input Properties...

The *Input Properties* form appears.

Material Name:	**steel**

OK

Select Members:	**Solid 1**

Add

Apply

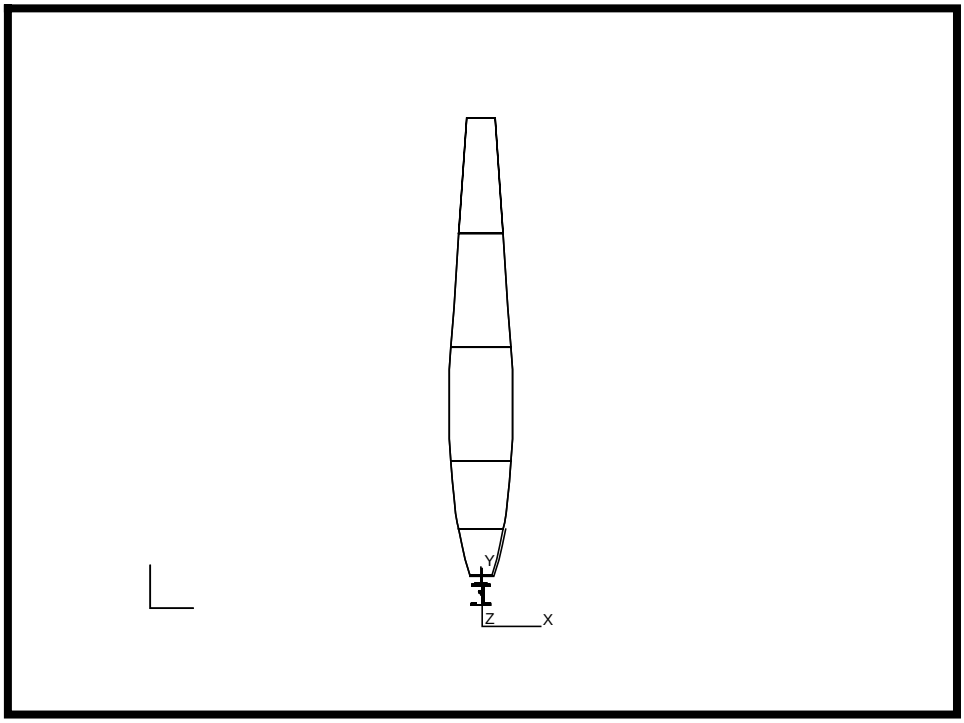
Create a user-defined coordinate frame at the base of the fin.

◆ **Geometry**

Action:	**Create**
Object:	**Coord**
Method:	**3Point**
Origin:	**[0 -2 0]**
Point on Axis 3:	**[0 -2 1]**
Point on Plane 1-3:	**[1 -2 0]**

Apply

The new coordinate frame can be seen in the viewport.



Determine Mass Properties

6. Determine the mass properties for the solid model of the fin.

Tools/Mass Properties...

Action: **Show**

Dimension: **3D**

Define Region...

The *Region for Mass Properties* form appears.

Region: **Group**

Include: **Geometry**

Select Groups: **solid**

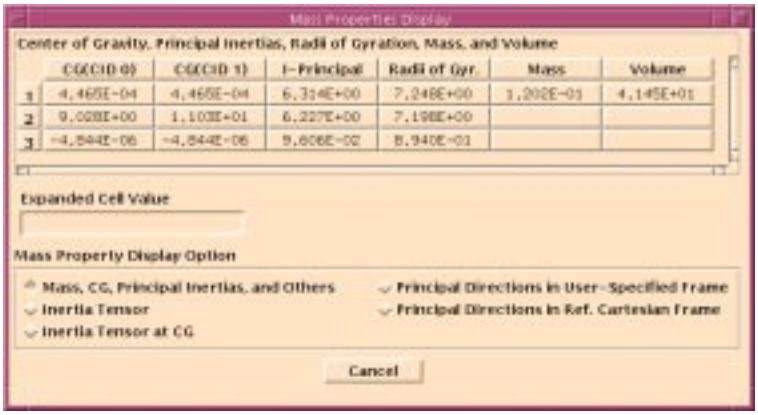
OK

Relative to Coordinate Frame: select newly created **Coord 1**

■ **Create Princ. Coord. Frame**

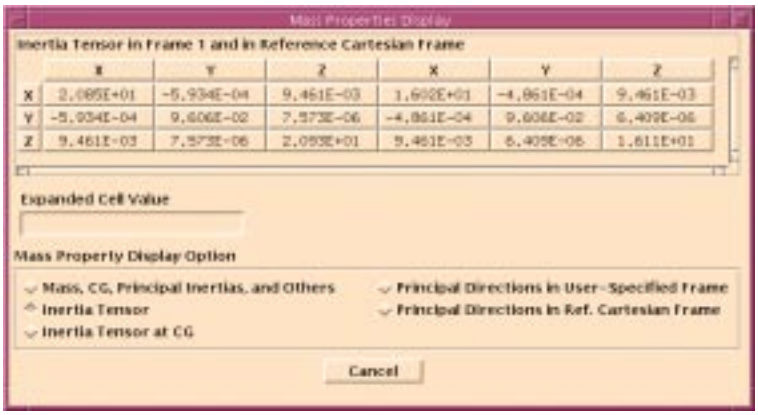
Apply

The *Mass Properties Display* form appears.



View the inertia tensor.

Mass Property Display Option: ◆ **Inertia Tensor**



Cancel

Close the Mass Properties Form.

Cancel

7. Create a group named **fin_2d**. This group will contain a 2D projection of the solid model.

Group/Create...

Action: **Create**

New Group Name: **fin_2d**

■ **Make Current**

❒ **Unpost All Other Groups**

Apply

Cancel

8. Create a 2D projection of the solid fin geometry.

Create a Projection

◆ **Geometry**

Action: **Create**

Object **Curve**

Method **Project**

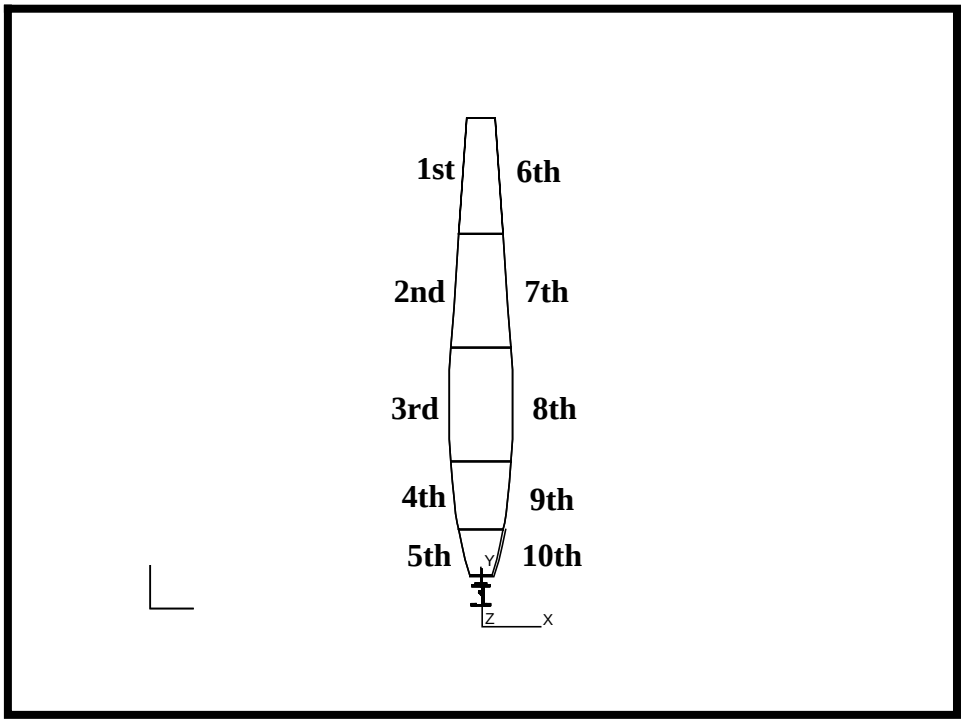
Project Onto: **Plane**

Option: **Normal to Plane**

❒ **Auto Execute**

Select the curves that define the left and right sides of the blade. Click on the **Edge** icon to select the edge of the solid, and shift-click to select the edges in the following order:

Edge



Curve List:	see figure above
Plane List:	**Coord 0.3**

To select **Coord 0.3** click on the **Axis 3** icon

Then hold down the middle mouse button to rotate the coordinate axis until you can screen select the **Z axis**.

Apply

9. Unpost all other groups to view the projection only

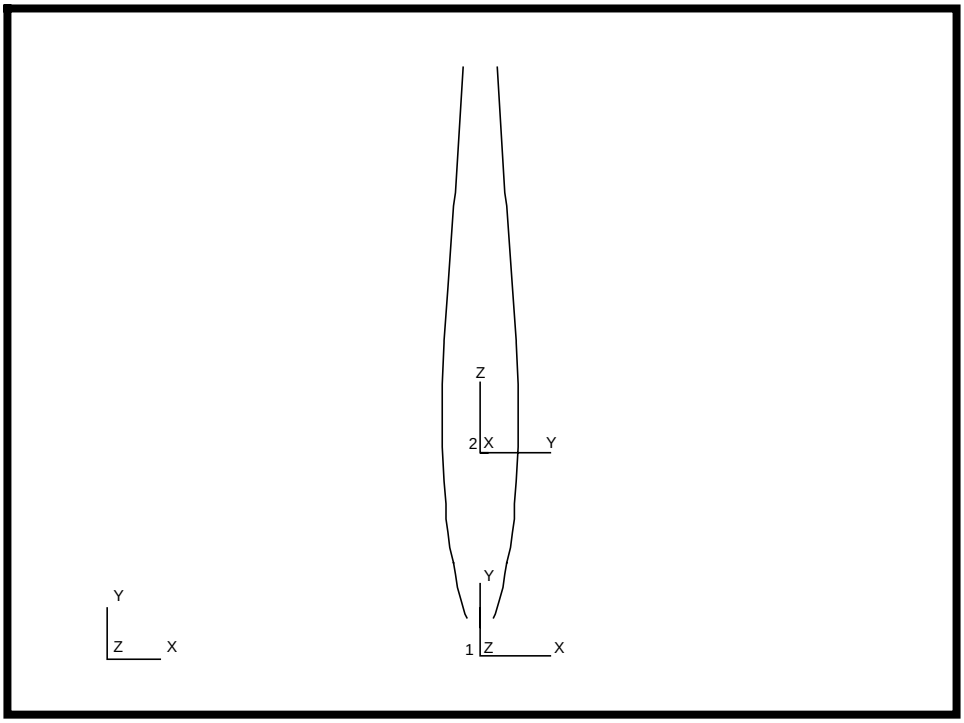
Group/Post...

Action:	**Post**
Select Groups to Post:	**fin_2d**

Apply

Cancel

The projected curves appear as follows.



10. Create a surface using the two projected curves.

Action:	**Create**
Object:	**Surface**
Method:	**Curve**
Option:	**2 Curve**

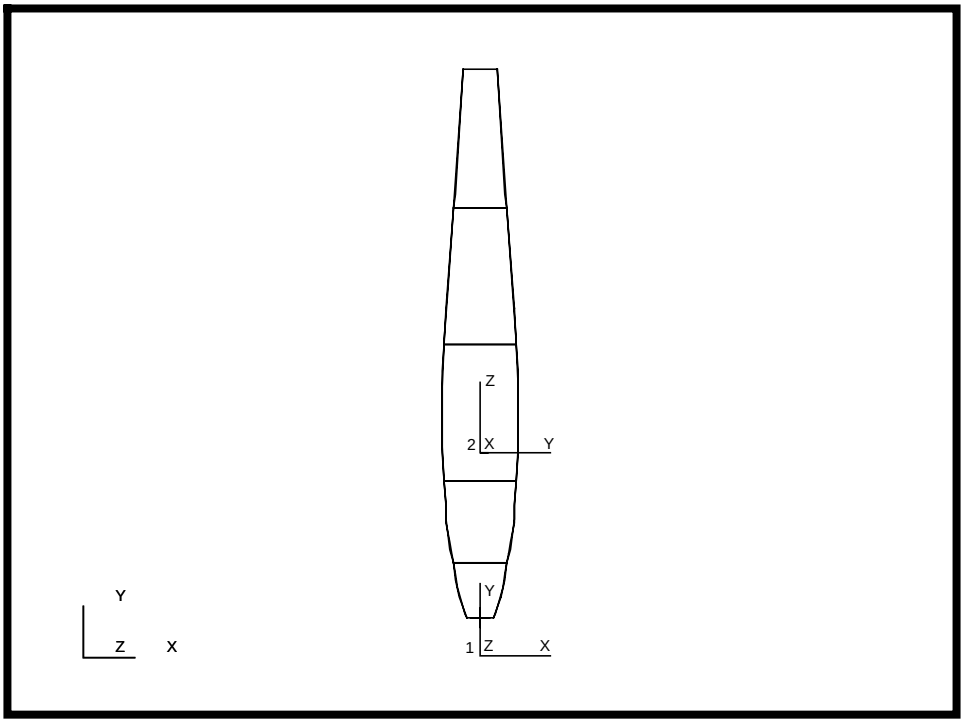
To select the curves, be sure to use the following entity select icon:

Curve

Starting Curve List:	select all curves on left side
Ending Curve List:	select all curves on right side

The function autoexecutes.

The surface should appear as follows:.



11. Create a field to represent the thickness of the surface.

Create a Field

◆ **Fields**

Action:	**Create**
Object:	**Spatial**
Method:	**PCL Function**
Field name:	**fin_thickness**
Field Type:	◆ **Scalar**
Coordinate System Type:	◆ **Real**
Coordinate System	**Coord 0**
Scalar Function (’X, ’Y, ’Z):	**0.3*abs(cosr(0.5*’X))**

Apply

12. Create a property set for the 2D surface model.

Create a Property Set

◆ **Properties**

Action:	**Create**

Dimension: **2D**

Type: **Shell**

Property Set Name: **fin_2d_shell**

Input Properties...

The *Input Properties* form appears.

Material Name: **m:steel**

Thickness: **f:fin_thickness**

OK

Select Members: **Surface 1:5**

Add

Apply

13. Check the mass properties for the surface.

Determine Mass Properties

Tools/Mass Properties...

Action: **Show**

Dimension: **3D**

Define Region...

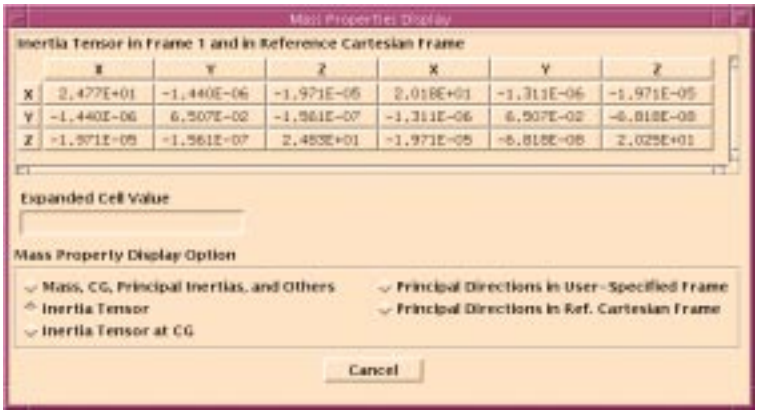
The *Region for Mass Properties* form appears.

Select Groups: **fin_2d**

OK

Apply

The *Mass Properties Display* form appears. Compare the inertia tensor to the one calculated for the solid.



Cancel

Check the *Warnings* form. Note that the curves and points were excluded from the calculation.

OK

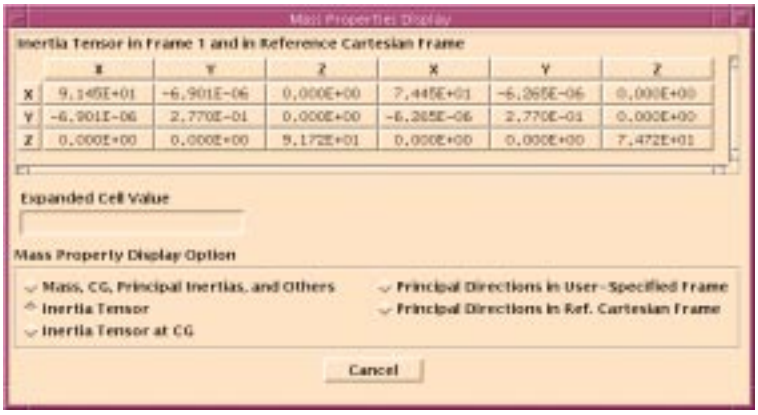
14. Calculate mass properties again using the 2D plane stress. The thickness of the blade is set to 1 by default.

Action: **Show**

Dimension: **2D Plane Stress**

Apply

The *Mass Properties Display* form appears. Compare the inertia tensor to the ones previously generated.



Cancel

Cancel

Close the warning form.

OK

Create a Surface Mesh

15. Tet mesh the base of the solid model of the fin. The geometry of the blade is very simple to mesh, so it will not be discussed. Besides, the base of the fin is usually where the focus of the analysis is directed, as that is usually the region of highest stresses.

    Post the **solid** group and unpost all other groups.

    Group/Post...

    Select Groups to Post: **solid**

    Apply

    Cancel

    Create a plane at the base of the blade to cut the solid.

    ◆ Geometry

    Action: **Create**

    Object: **Plane**

    Method **Vector Normal**

    Plane Offset Distance: **0.35**

Vector List: **Coord 0.2**

Break the solid using the plane.

Action: **Edit**

Object: **Solid**

Method: **Break**

Option: **Plane**

❒ **Auto Execute**

Solid List: **Solid 1**

Break Plane List: **Plane 1**

Apply

A message form appears. Do not delete the original solids.

No For All

Clean up the display using the following toolbar icon:

Refresh Graphics

Create a new group named **solid_with_tetmesh** and add the base of the newly trimmed solid to it. Unpost all other groups.

Group/Create...

Action: **Create**

New Group Name: **solid_with_tetmesh**

■ **Unpost All Other Groups**

Entity Selection **Solid 2** (the base of the fin)

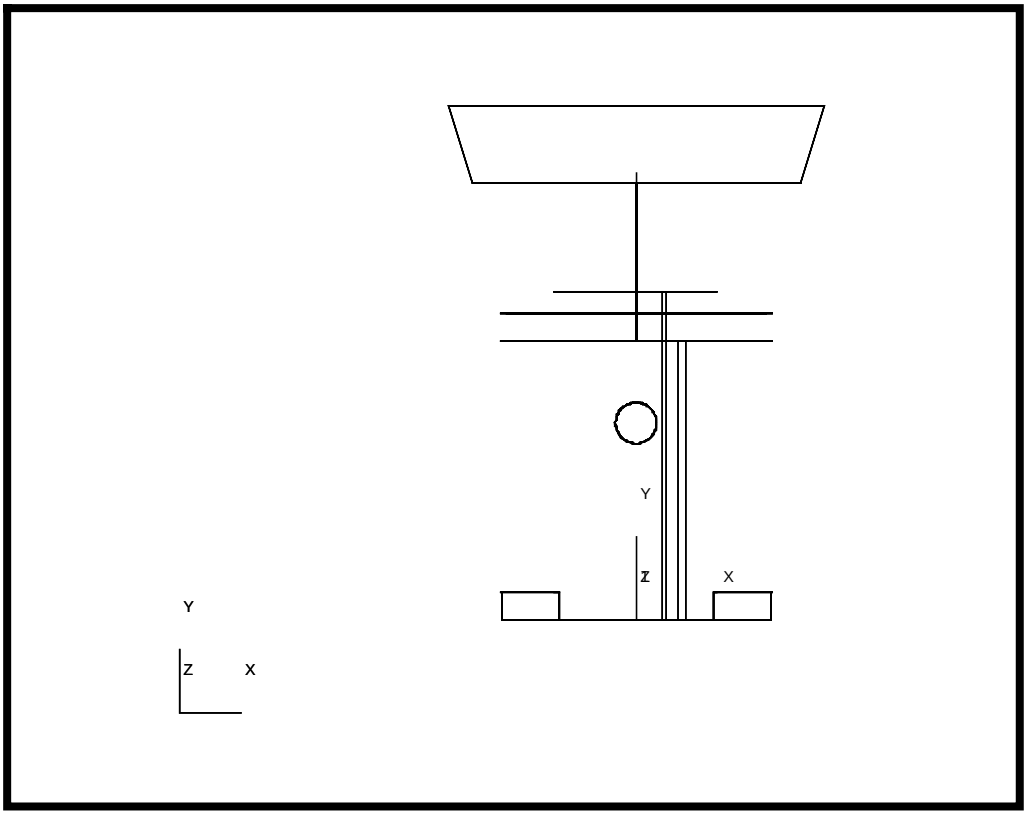
Apply

Cancel

Now, zoom in on the base of the fin using the following toolbar icon:

View Corners

The new solid should appear as follows.



Now, mesh the solid base.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Method	**Solid**
Global Edge Length:	**0.31**
Mesher:	◆ **Tetmesh**
Input List:	select the base solid

Apply

16. Conduct a Mass Properties calculation on the base of the fin.

First, assign properties to the base of the solid.

◆ **Properties**

Action: **Create**

Dimension: **3D**

Type: **Solid**

Property Set Name: **base_props**

Input Properties...

The *Input Properties* form appears.

Material Name: **steel**

OK

In order to select the tet elements of the new mesh, use the following entity select icon:

 **Solid Element**

Select Members: select all solid elements

Add

Apply

Now, create a group called **mesh**.

Group/Create...

Action: **Create**

New Group Name: **mesh**

■ **Unpost All Other Groups**

Entity Selection **Add All FEM**

Apply

17. Perform a Mass Properties calculation on the base.

Tools/Mass Properties...

Action: **Show**

Dimension: **3D**

Define Region...

The *Region for Mass Properties* form appears.

Include: **FEM**

Select Groups: **mesh**

OK

■ **Create Princ. Coord. Frame**

Apply

This ends the exercise.