5. SEDAN GEOMETRY—VIRTUAL CLEANUP

In this tutorial you will import an IGES file containing the geometry for a sedan automobile, clean up the geometry, and mesh it with triangles and tetrahedra.

In this tutorial you will learn how to:

- Import an IGES file

- Specify the way in which the geometry will be colored

- Connect edges, using a manual and an automatic method

- Merge faces

- Create a triangular surface mesh

- Mesh a volume with a tetrahedral mesh

- Prepare the mesh to be read into FLUENT 5/6

5.1 Prerequisites

This tutorial assumes you have worked through Tutorial 1 and, therefore, that you are familiar with the GAMBIT GUI.

5.2 Problem Description

The problem to be considered is shown schematically in Figure 5-1; it is the external body of a luxury sedan. You will generate a mesh on the outside of the car body; therefore, you will create a brick around the sedan to represent the flow domain.
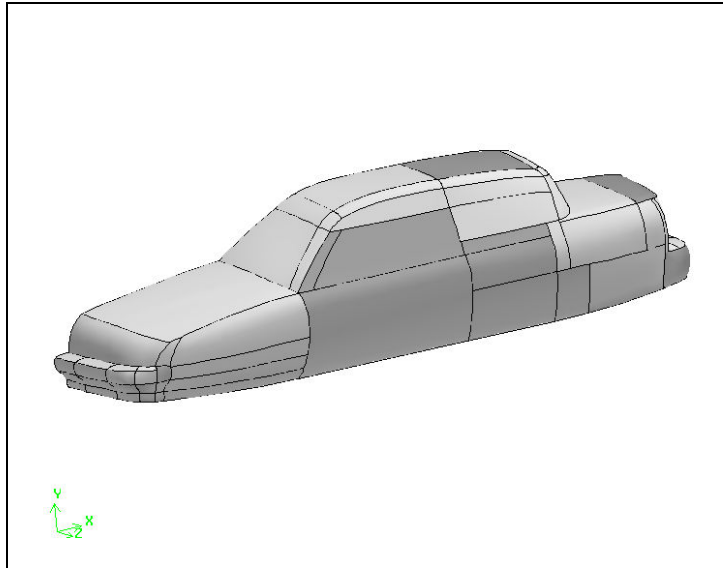

Figure 5-1: Sedan geometry

5.3 Strategy

In this tutorial, you will create a fully unstructured tetrahedral mesh around a car-body geometry imported as an IGES file. This tutorial illustrates the steps you would typically follow to prepare an imported CAD geometry for meshing. The imported geometry is "dirty"—that is, there are gaps between some of the surfaces that make it unsuitable for creating a CFD mesh. After examining the raw imported geometry to identify its problems, such as unconnected edges, you will clean up the geometry using the tools available in GAMBIT.

Most of the gaps can be fixed automatically either during mesh import or subsequently by means of the "connect edge" command. The original CAD geometry is not modified during the fixing process; the modifications required to eliminate the gaps are made using "virtual" geometry, which lies on top of the "real" geometry. Some edges in the original geometry are very short and will be eliminated using the "vertex connect" command. Other edges are not automatically connected, because they are farther apart than the specified tolerance. You will connect such edges manually.

The imported geometry includes a number of small surfaces, the edges of which may unnecessarily constrain the mesh generation process. Using the "merge faces" command, GAMBIT allows you to easily combine these surfaces prior to meshing. You can then have GAMBIT automatically create a triangular mesh on the car body.

Since the imported geometry consists only of the car body, you need to create a suitable domain around the car in order to conduct a CFD analysis (this is loosely equivalent to placing the car in a wind tunnel). The remainder of the tutorial shows how to add a real box around the car body, use virtual geometry to create some missing faces, and finally stitch all faces together into a single volume. This volume can then be meshed (without any decomposition) using a tetrahedral meshing scheme.

5.4 Procedure

1. Copy the file

   path/Fluent.Inc/gambit2.*x*/help/tutfiles/sedan.igs

   (where 2.*x* is the GAMBIT version number) from the GAMBIT installation area in
   the directory *path* to your working directory.

2. Start GAMBIT.

Step 1: Select a Solver

1. Choose the solver from the main menu bar:

   Solver → FLUENT 5/6

   *The choice of a solver dictates the options available in various forms (for
   example, the boundary types available in the* **Specify Boundary Types** *form). For
   some systems,* **FLUENT 5/6** *is the default solver. The solver currently selected is
   shown at the top of the* GAMBIT *GUI.*

Step 2: Import the IGES File As-Is

File → Import → IGES ...

This command sequence opens the **Import IGES File** *form.*

1. Click on the **Browse...** button.

    This action opens the **Select File** *form.*
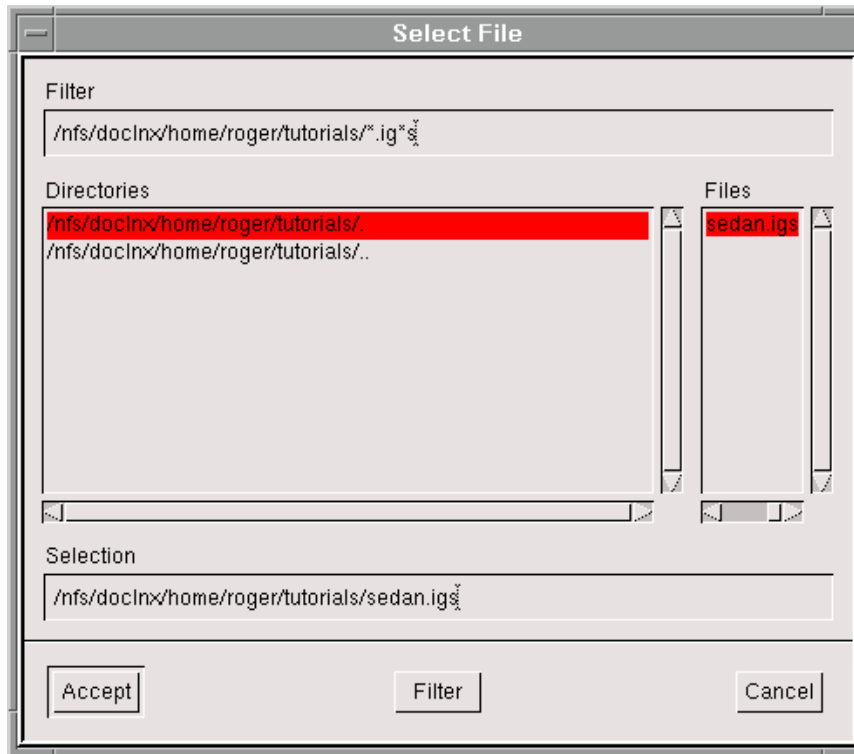
    a) Select `sedan.igs` in the **Files** list.

    b) Click Accept in the **Select File** form.

2. On the **Import IGES File** form, unselect the **Make Tolerant** option, and click **Accept**.

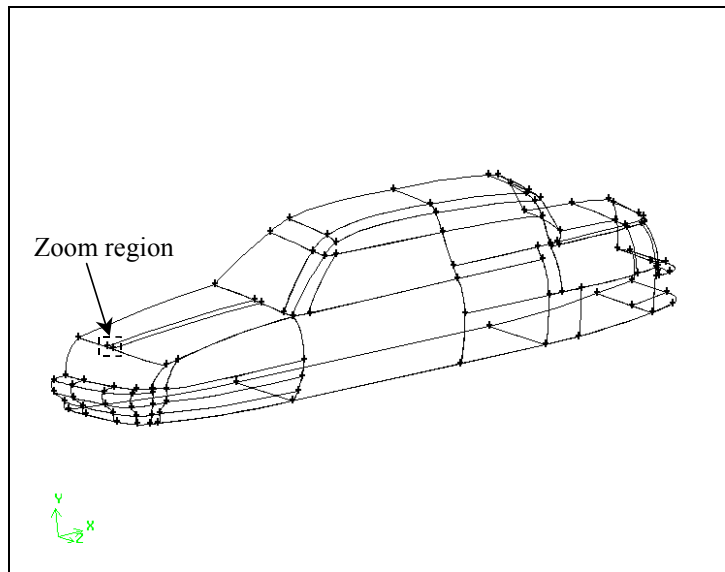   *The IGES file for the sedan body will be read into* GAMBIT *(see Figure 5-2).*

Figure 5-2: Imported sedan body

3. Click the **SPECIFY COLOR MODE** command button ![icon] in the **Global Control** toolpad to change to the graphics display to connectivity-based coloring.

    The **SPECIFY COLOR MODE** *command button will change to* ![icon]. *When GAMBIT is in the connectivity display mode, the model is displayed with colors based on connectivity between entities rather than based on entity types.*

    In this case, the colors of all edges in the graphics window will change to orange, indicating that the faces are not connected to each other—that is, that there are gaps between the edges that bound the faces.

4. *Ctrl*-drag the mouse in the graphics window to create the zoom-region box shown in Figure 5-2, above (at the front of the sedan hood). When you release the mouse button, GAMBIT zooms in on the region.

    Figure 5-3 shows the zoomed region, illustrating an example of the gaps that exist between the faces that comprise the surface of the sedan.
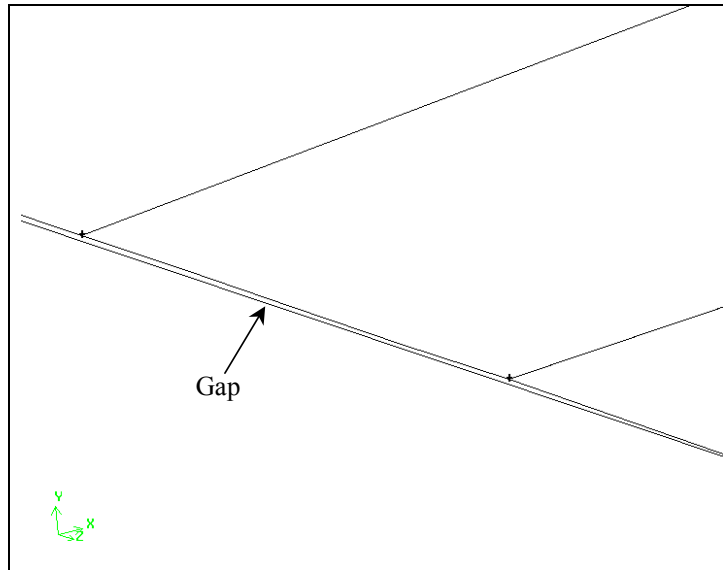
5-7

Figure 5-3: Zoomed region showing unconnected faces on sedan hood

In addition to gaps such as those shown in Figure 5-3, the geometry imported as-is from the IGES file contains a number of small edges that can inhibit meshing of the model. GAMBIT provides options on the **Import IGES File** *form that allow you to eliminate many of the gaps and short edges during import. The next step in this tutorial illustrates the use of one such option—***Virtual Cleanup***.*

Step 3: Reset and Import the IGES File Using Virtual Cleanup

The **Virtual Cleanup** *option on the* **Import IGES File** *form allows you to automatically eliminate many features, such as short edges and gaps between unconnected faces, during geometry import. By eliminating many such features during geometry import, you can greatly simplify the process of creating a meshable model.*

1. Reset GAMBIT.

    a) On the **Command** line, type `reset`, and press *Enter*.

        GAMBIT *deletes the current geometry from the graphics window and resets the GUI.*

2. Reselect the FLUENT 5/6 solver from the main menu bar.

    Solver → **FLUENT 5/6**

3. Open the **Import IGES File** form.

    File → **Import** → **IGES ...**

    This command sequence opens the **Import IGES File** *form.*

5-9

GAMBIT *retains the* **File Name** *selected in Step 2—that is,* sedan.igs.

4. Under **Import Options: Stand-alone Geometry**, select the No stand-alone edges check box.

This option instructs GAMBIT *not to read in any edges that do not belong to faces or volumes. Such edges can be deleted after the geometry has been read into* GAMBIT*, but this option eliminates the extra step.*

5. Set the connect tolerance to 10% of the shortest edge by selecting the **Virtual Cleanup** toggle and specifying the **Shortest Edge %** at 10.

    This substep invokes an automated sequence of connect operations that attempt to clean up the imported geometry after it is read into GAMBIT.

6. Click the **SPECIFY COLOR MODE** command button ![button] in the **Global Control** toolpad to change to the graphics display to connectivity-based coloring.

7. On the **Import IGES File** form, click **Accept**.

    The IGES file for the sedan body will again be read into GAMBIT. *This time, however,* GAMBIT *will perform virtual clean-up operations to eliminate gaps and many small edges. As the clean-up operation progresses, edges that are initially displayed as orange (with the connectivity color mode on) will turn blue. Figure 5-4 shows the sedan geometry after import and clean-up operations are complete.*
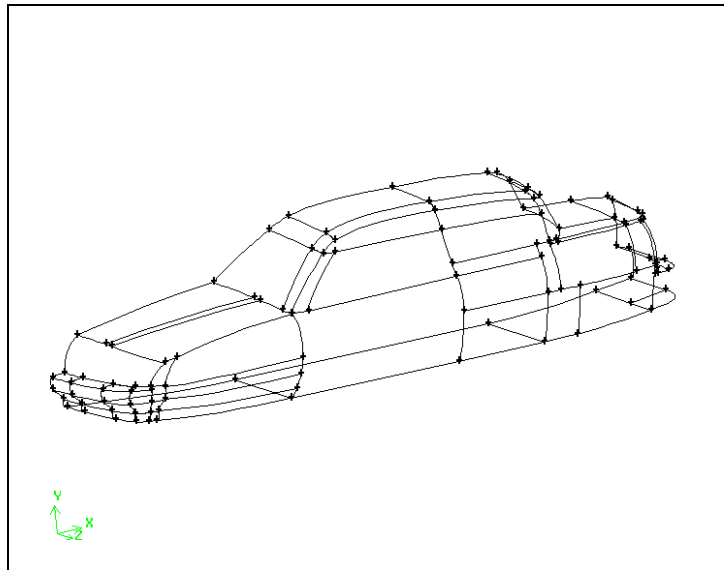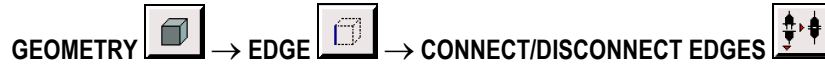

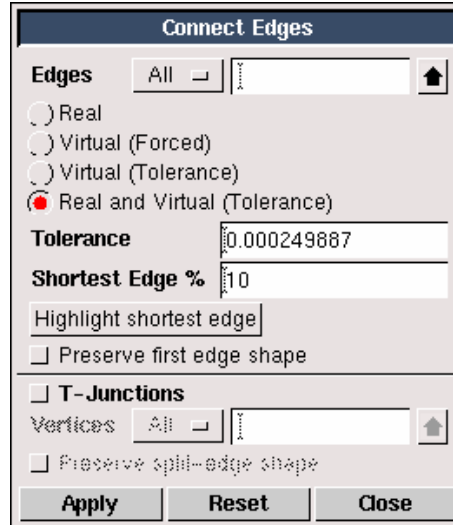Figure 5-4: Imported sedan body—with **Virtual Cleanup** option

Step 4: Eliminate Very Short Edges

*Even after the virtual clean-up operations, the imported IGES geometry is still some-
what "dirty"—that is, there are a few short edges and gaps between the faces that
need to be repaired. In this step, you will eliminate the short edges.*

1. Find the shortest edge.

    GEOMETRY 🔲 → **EDGE** 🔲 → **CONNECT/DISCONNECT EDGES** ⬍

    This command sequence opens the **Connect Edges** *form.*

    a) Select All from the option menu to the right of **Edges**.

    b) Select the Real and Virtual (Tolerance) option.

    c) Press the Highlight shortest edge button.

        GAMBIT *will highlight (in white) the shortest edge—along with its label—in
        the graphics window.*

    d) Zoom in near the highlighted edge by pressing the *Ctrl* key while using the mouse
        to drag a box around the edge.

        *Figure 5-5 shows the general area on the sedan that contains the shortest
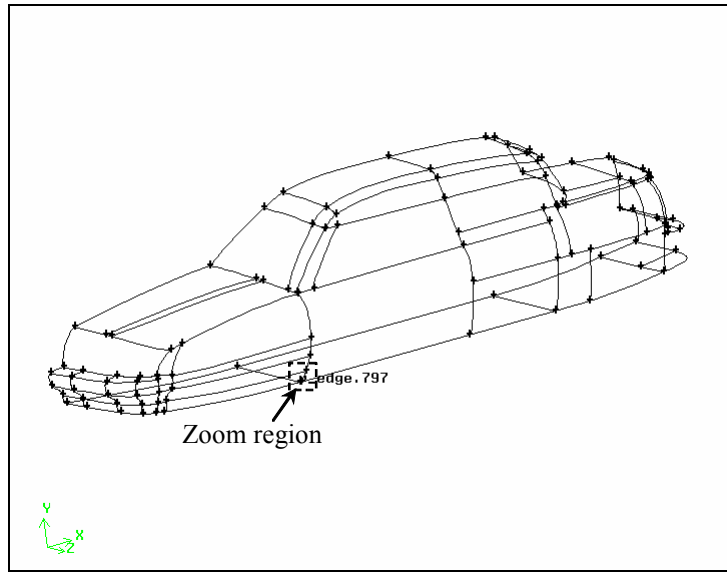   edge. Figure 5-6 shows a zoomed view of the edge.*

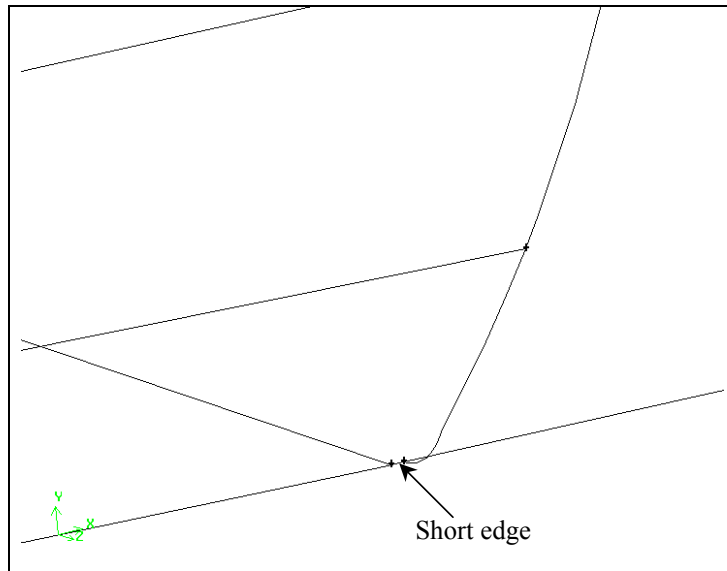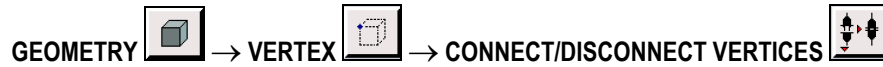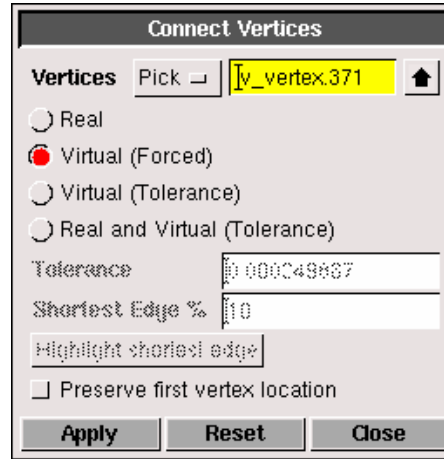Figure 5-5: Sedan—showing general area of shortest edge location

Figure 5-6: Sedan—showing zoomed area near shortest edge

2. Remove the shortest edge.

    GEOMETRY ⬛ → **VERTEX** ▭ → **CONNECT/DISCONNECT VERTICES** ⬛

    This command sequence opens the **Connect Vertices** *form.*

    Connect Vertices

    Vertices   Pick ⊐ | v_vertex.371 | ⬆

    ◯ Real
    🔴 Virtual (Forced)
    ◯ Virtual (Tolerance)
    ◯ Real and Virtual (Tolerance)

    Tolerance | 0.000049867 |
    Shortest Edge % | 10 |
    Highlight shortest edge
    ☐ Preserve first vertex location

    | **Apply** | **Reset** | **Close** |

    a) Select the Virtual (Forced) option.

    b) Pick the two endpoint vertices on the shortest edge.

    c) Click **Apply**.

        When GAMBIT *attempts to connect these two vertices, an error message is generated stating that connecting these two vertices can cause invalid geometry. The geometry is protected from such operations by means of a default setting.*

    d) Open the **Edit Defaults** form (select **Defaults...** from the **Edit** menu on the main menu bar), and change the value of the GEOMETRY.VERTEX.CONNECT_REMOVE_ SHORT_EDGE variable to 1.

    e) Repeat step (c).

        This time, GAMBIT *connects the vertices.*

    f) Click the **FIT TO WINDOW** command button ⬛ at the top left of the **Global Control** toolpad to see the full sedan in the graphics window.

g) Select the Virtual (Tolerance) option to activate the Highlight shortest edge button on the **Connect Vertices** form.

h) Click the Highlight shortest edge button and repeat steps (a), (b), and (c) to eliminate the next shortest edge (see Figure 5-7).
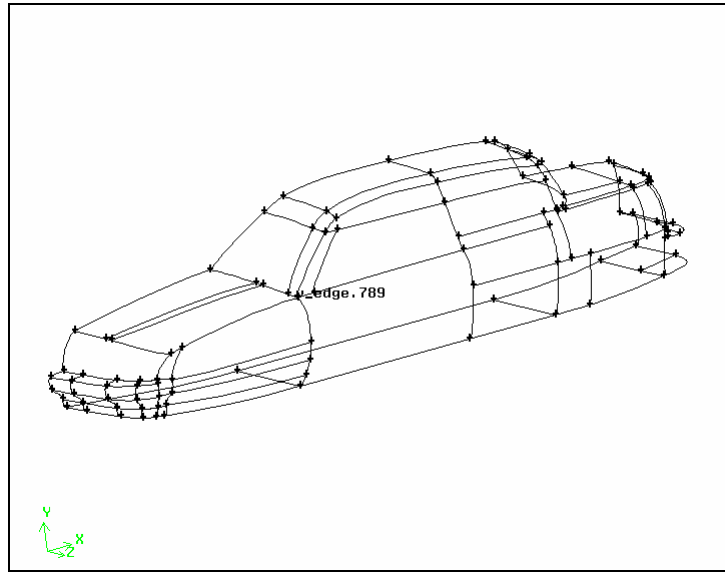

Figure 5-7: Sedan—showing location of next shortest edge

By eliminating the two shortest edges in the model, you ensure that all edge meshing intervals are of reasonable size, thereby reducing the possibility of creating highly distorted elements during meshing.
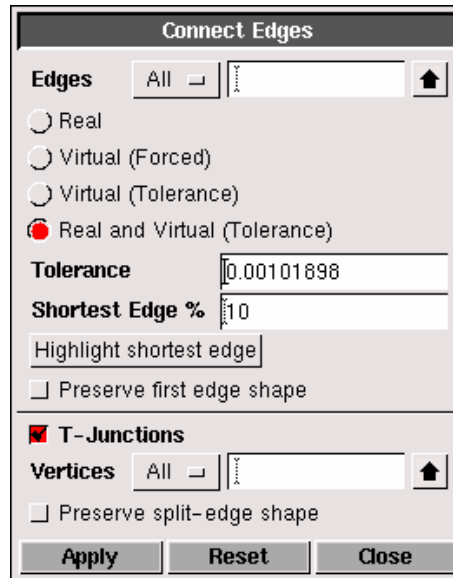
Step 5: Automatically Connect All Remaining "Duplicate" Edges

The imported IGES geometry is still "dirty"—that is, there are a few gaps remaining between the faces that make it unsuitable for creating a mesh. In this step, you will "clean up" the geometry automatically using the **Connect Edges** *operation.*

1. Connect all edges in the geometry that are less than a specified tolerance apart using an automatic method.
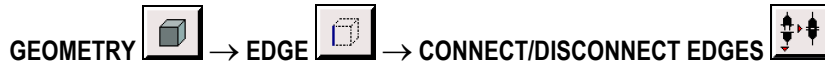
   GEOMETRY [ ] → **EDGE** [ ] → **CONNECT/DISCONNECT EDGES** [ ]

   This command sequence opens the **Connect Edges** *form.*

   a) Select All from the option menu to the right of **Edges**.

   b) Select the Real and Virtual (Tolerance) option.

   > *You want* GAMBIT *to connect all real and virtual edges that are within a tolerance distance of each other.*

   c) Retain the default value of 10 for the **Shortest Edge %**.

   d) Select the **T-Junctions** option.

This option ensures that edges that do not match up correctly will be connected. GAMBIT *will perform edge splits and then reconnect the geometry; an example is shown in Figure 5-8.*
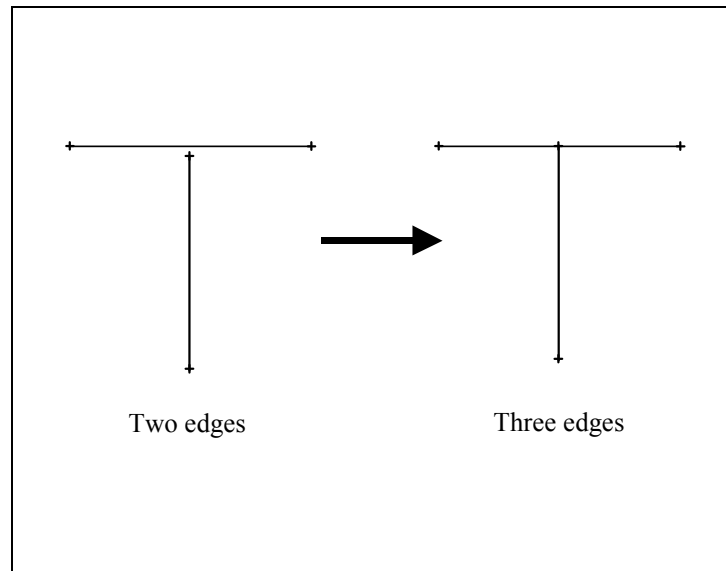

Figure 5-8: Connecting edges

e) Click **Apply**.

> *A few more edges turn blue in the graphics window as they are connected.*
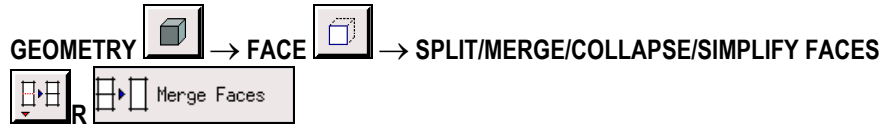
> ! *The edges on the symmetry plane will remain orange because they do not have any other edges with which they can be connected.*

2. Click the **FIT TO WINDOW** command button ![button] at the top left of the **Global Control** toolpad to see the full sedan in the graphics window.
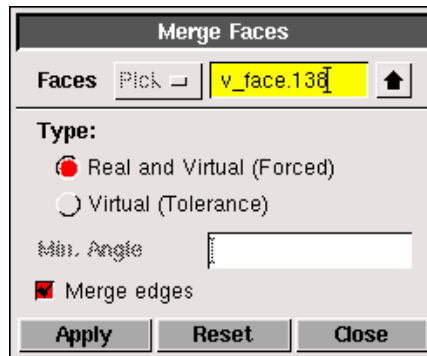
Step 6: Merge Faces

In many cases, the IGES model contains more detail than you need for meshing. The imported geometry for the sedan includes a number of small faces, the edges of which may constrain the mesh generation process unnecessarily. In GAMBIT*, you can merge faces together prior to meshing.*

1. Merge some of the faces on the sedan hood.

GEOMETRY [■] → **FACE** [□] → **SPLIT/MERGE/COLLAPSE/SIMPLIFY FACES** [▨]**R** [▨] Merge Faces

This command sequence opens the **Merge Faces** *form.*

Merge Faces
Faces Pick ▢ │ v_face.138 │ ▲
Type:
● Real and Virtual (Forced)
○ Virtual (Tolerance)
Min. Angle [ ]
☑ Merge edges
Apply   **Reset**   **Close**

a) Under **Type**, select Real and Virtual (Forced).

b) Zoom in to the hood of the sedan by holding down the *Ctrl* key on the keyboard while dragging a box around the hood of the car with the left mouse button.

c) Select the three faces on the top of the hood as shown in Figure 5-9.

d) Retain the Merge Edges option to facilitate geometry cleanup during merging.

e) Click **Apply** to accept the selected faces and merge them into one face, as shown in Figure 5-10.
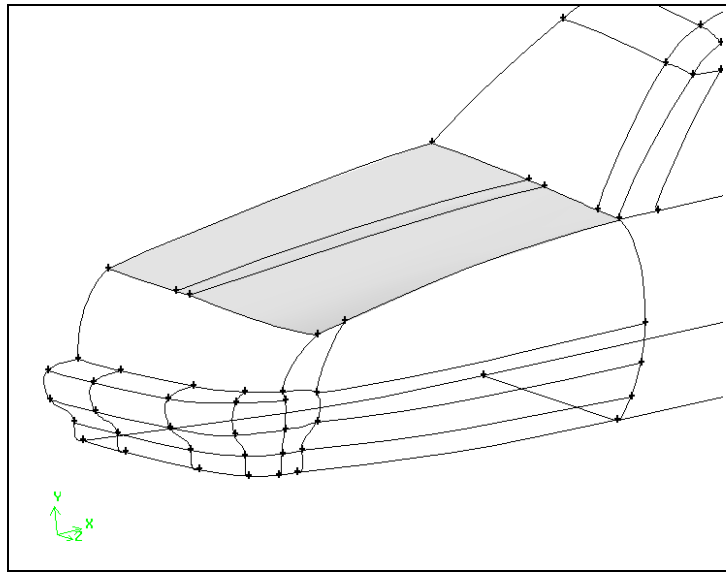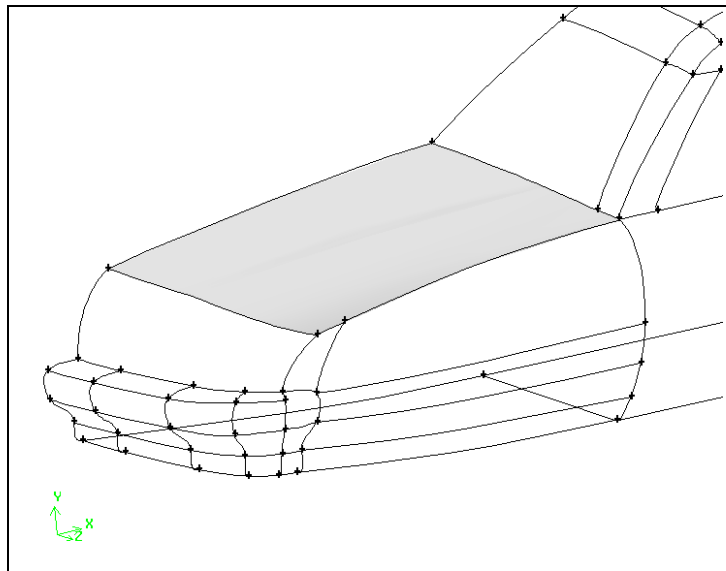
Figure 5-9: Three faces on hood of sedan

Figure 5-10: Three faces merged on hood of sedan

2. Merge four faces on the trunk of the car (see Figure 5-11) using the method described above. The merged faces are shown in Figure 5-12.
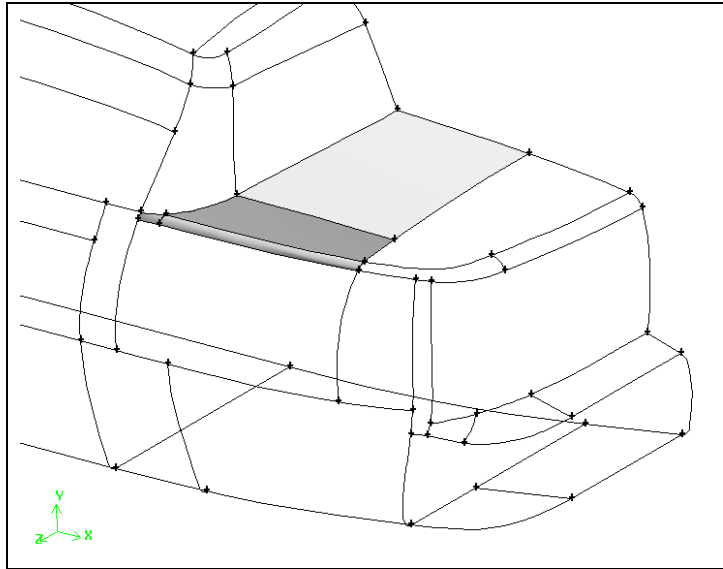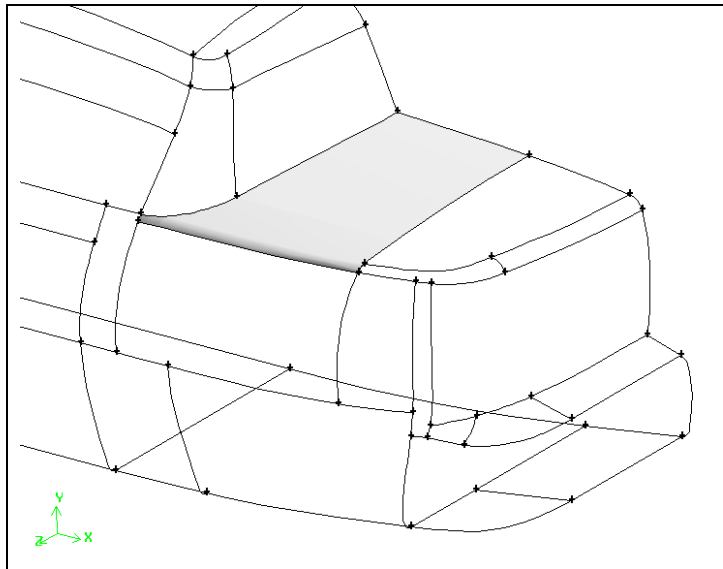

Figure 5-11: Four faces on trunk of sedan

Figure 5-12: Four faces merged on trunk of sedan

3. Merge three faces near the rear end of the trunk of the car (see Figure 5-13) using the method described above. The merged faces are shown in Figure 5-14.
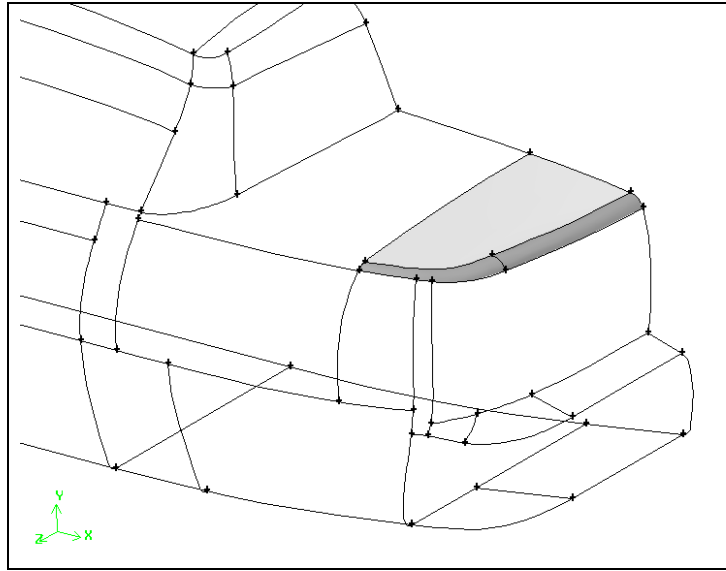

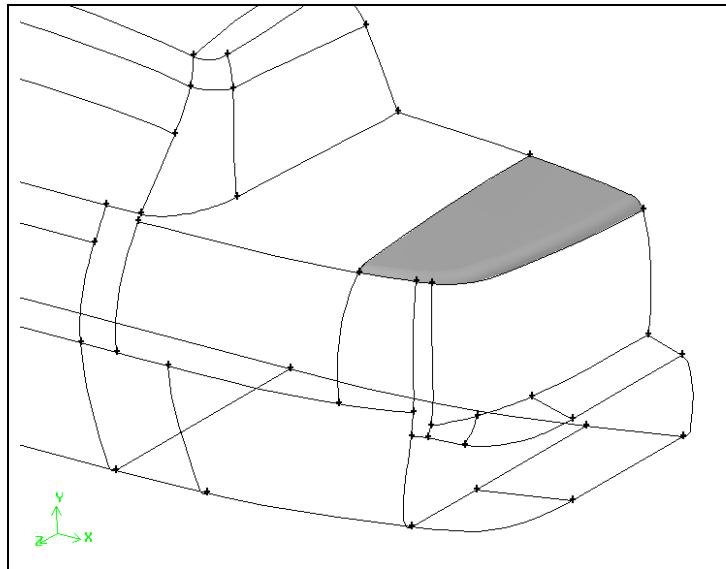Figure 5-13: Three faces near rear end of trunk

Figure 5-14: Three faces merged near rear end of trunk

4. Click the **FIT TO WINDOW** command button ![] at the top left of the **Global Control** toolpad to see the full sedan in the graphics window.

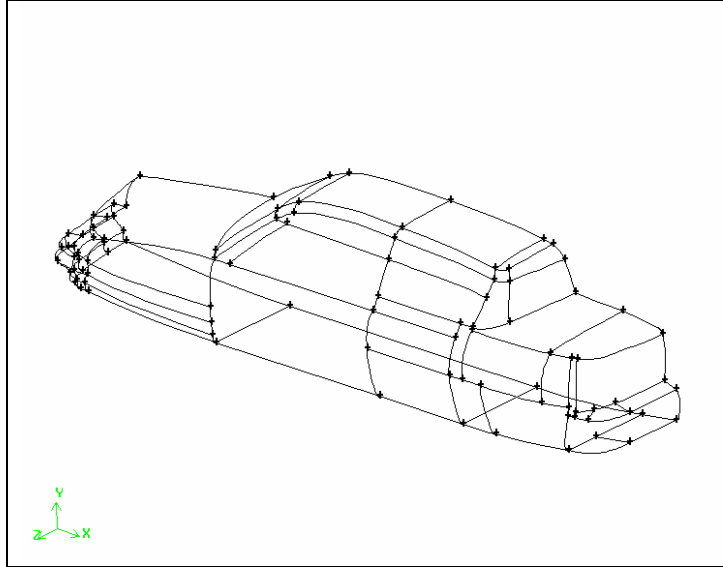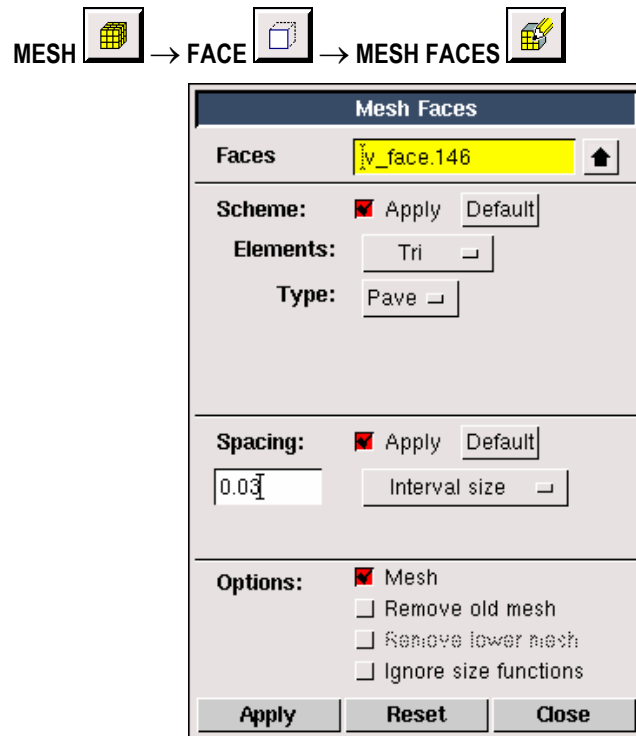> *The top portion of the trunk should now consist of two large faces, as shown in Figure 5-15.*

Figure 5-15: Merged faces on sedan

Step 7: Mesh Faces on Car Body

1. Create a surface mesh on the faces of the car body.

MESH ⊞ → FACE ▢ → MESH FACES 📝

Mesh Faces
Faces   v_face.146   ▲
Scheme: ☑ Apply   Default
Elements: Tri ⌐
Type: Pave ⌐
Spacing: ☑ Apply   Default
0.03   Interval size ⌐
Options: ☑ Mesh
☐ Remove old mesh
☐ Remove lower mesh
☐ Ignore size functions
Apply   **Reset**   **Close**

a) Select all the faces on the car body by holding down the *Shift* key and using the left mouse button to drag a box around the whole geometry in the graphics window.

   !   *It may take a while for* GAMBIT *to select all the faces.* GAMBIT *analyzes each face to determine suitable meshing schemes. You should wait until all the edges turn red before going on to the next step.*

b) Select Tri from the **Elements** option button under **Scheme**.

   GAMBIT *automatically selects the* **Type:**Pave *option. For more information on face meshing schemes, see the* GAMBIT *Modeling Guide.*

c) Enter an Interval size of 0.03 under **Spacing** and click the **Apply** button at the bottom of the form.

GAMBIT *will mesh the car body surfaces. A portion of the mesh is shown in Figure 5-16.*
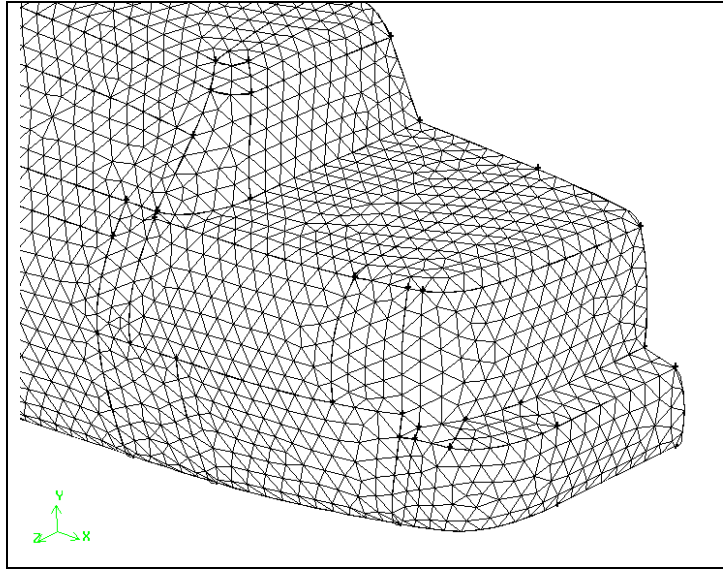

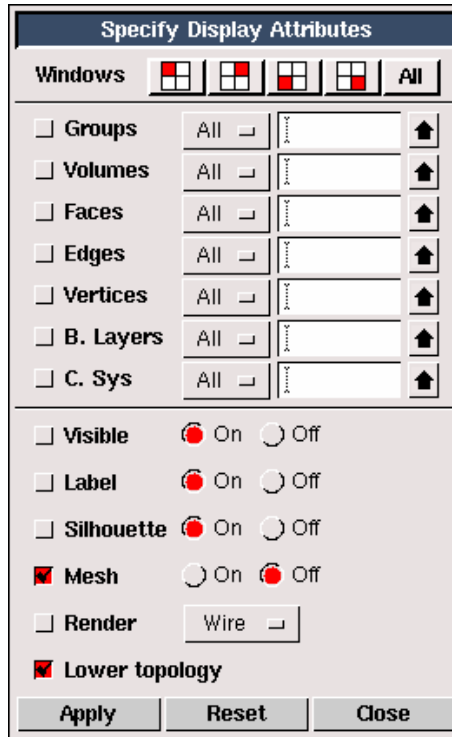Figure 5-16: Surface mesh on rear of car body

2. Remove the mesh from the display.

   ! *This will make it easier to see what to do in the next steps. The mesh is not deleted, just removed from the graphics window.*

   a) Click the **SPECIFY DISPLAY ATTRIBUTES** command button at the bottom of the **Global Control** toolpad.

      This action opens the **Specify Display Attributes** *form.*

b) Select the Off radio button to the right of **Mesh** near the bottom of the form.

    GAMBIT *will automatically select the* **Mesh** *check box.*

c) Click **Apply** and close the form.
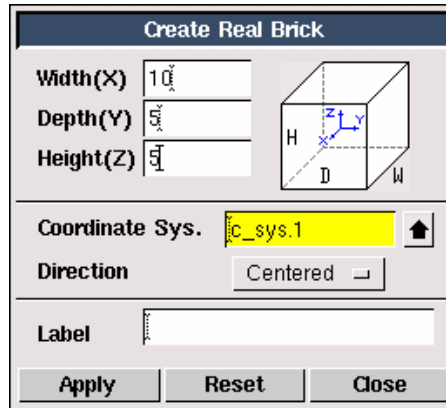
    The mesh will be removed from the graphics window.

Step 8: Create a Brick Around the Car Body

1. Create a brick.

   GEOMETRY [icon] → **VOLUME** [icon] → **CREATE VOLUME** [icon]

   This command sequence opens the **Create Real Brick** *form.*

   a) Enter a value of 10 for the **Width** of the brick.

   b) Enter 5 for the **Depth** and 5 for the **Height**.

   c) Select Centered from the option menu to the right of **Direction**.

   d) Click **Apply**.

2. Click the **FIT TO WINDOW** command button [icon] at the top left of the **Global Control** toolpad to see the full sedan and the brick just created in the graphics window.

3. Move the brick to the desired location relative to the sedan.

   GEOMETRY [icon] → **VOLUME** [icon] → **MOVE/COPY/ALIGN VOLUMES** [icon]

   This command sequence opens the **Move / Copy Volumes** *form.*

a) *Shift*-left-click the brick in the graphics window.

b) Select **Move** (the default) under **Volumes** in the **Move / Copy Volumes** form.

c) Select Translate (the default) under **Operation**.

d) Enter (0, 2.5, 2.5) under **Global** to move the brick 2.5 units in the *y* direction and 2.5 units in the *z* direction.

> *Note that* GAMBIT *automatically fills in the values under* **Local** *as you enter values under* **Global**.

e) Click **Apply**.

4. Click the **FIT TO WINDOW** command button [icon] at the top left of the **Global Control** toolpad to see the full sedan and the brick in the graphics window.

The brick and sedan are shown in Figure 5-17.
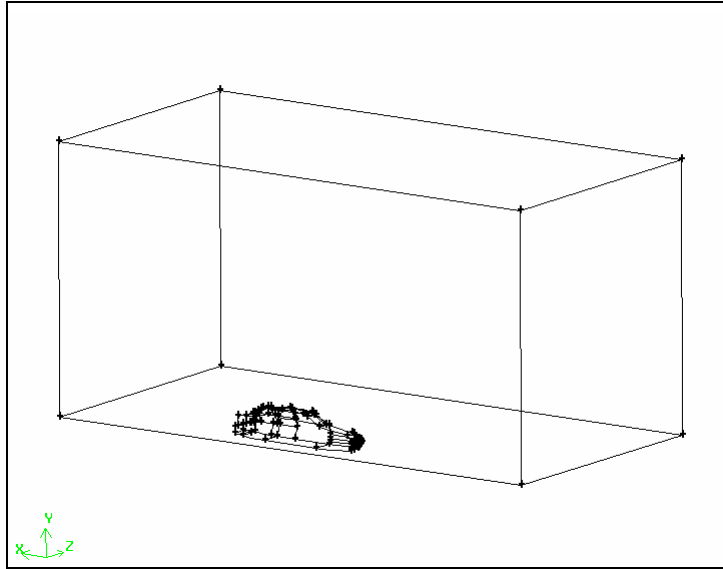
Figure 5-17: Brick and sedan

Step 9: Remove Unwanted Geometry

You cannot simply subtract the car from the brick to produce the flow domain around the car, because you used "virtual geometry" to clean up the car body and GAMBIT cannot perform Boolean operations on virtual geometry. Instead, you must "stitch together" a virtual volume from the virtual faces of the car and the real faces of the brick. To do this you will delete the volume of the brick, leaving the lower geometry (the faces) behind. In the next steps, you will create virtual edges and faces.

1. Delete the volume of the brick, leaving the faces behind.

    GEOMETRY ⬚ → **VOLUME** ⬚ → **DELETE VOLUMES** 🧽

    This command sequence opens the **Delete Volumes** *form.*

Delete Volumes
Volumes Pick ⊣ │ volume.1 │ ⬆
☐ **Lower Geometry**
Apply   **Reset**   **Close**
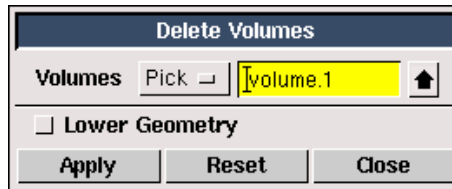
    a) *Shift*-left-click the brick in the graphics window.

    b) Unselect the **Lower Geometry** option and click **Apply**.

        The brick volume will be deleted, but all its components (faces, edges, and vertices) will remain in the geometry, because you deselected the **Lower Geometry** *option.*

Step 10: Create Straight Edges on the Symmetry Plane

In this step, you will create two straight edges that will be used in the next step to create faces on the symmetry plane.

1. Split the bottom edge of the symmetry plane into three sections.

GEOMETRY [▢] → **EDGE** [▢] → **SPLIT/MERGE EDGES** [▢]

This command sequence opens the **Split Edge** *form.*

a) Retain the **Type:**Real connected option.

b) Select **Split With** Point (the default).

> *You will split the edge by creating a point on the edge and then using this point to split the edge.*

c) Use the *Ctrl* key and the left mouse button to zoom in to the sedan and the line at the bottom of the symmetry plane, similar to the view shown in Figure 5-18.

d) Select the edge at the bottom of the symmetry plane in the graphics window.

e) Enter a **U Value** of 0.64 in the **Split Edge** form and click **Apply**.

The vertex needs to be close to the front of the sedan. A **U Value** *of* 0.64 *will place the vertex in the correct position, but it is the position relative to the sedan that is important, not the exact* **U Value**.

The edge is split into two parts and a vertex is created near the front bumper of the sedan, as shown in Figure 5-18.

f) Select the longer edge of the two edges just created in the graphics window.

g) Enter a **U Value** of 0.57 in the **Split Edge** form and click **Apply**.

Again, the position of the vertex relative to the sedan is more important than the exact **U Value**.

The edge will be split and a second vertex created near the rear bumper of the sedan, as shown in Figure 5-18.
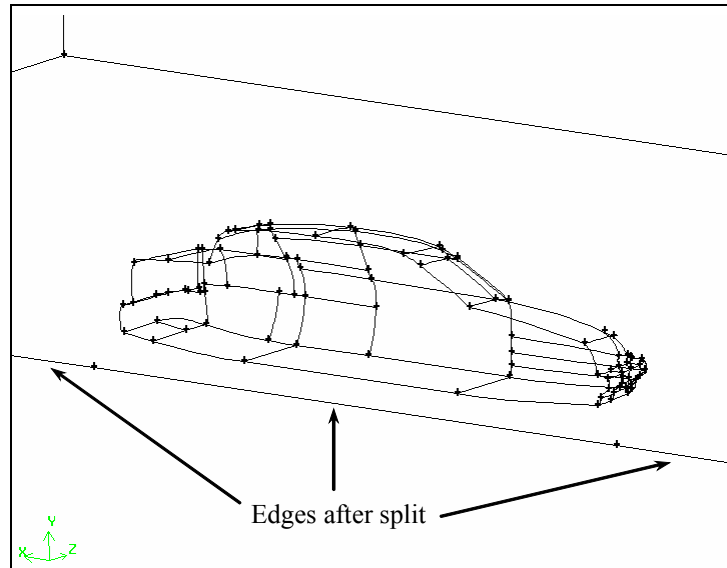
Figure 5-18: Bottom edge of symmetry plane is split into three edges

2. Create straight edges between the two points just created and two points on the sedan.

GEOMETRY [□] → **EDGE** [□] → **CREATE EDGE** [□]

This command sequence opens the **Create Straight Edge** *form.*

Create Straight Edge
Vertices [v_vertex.293] ⬆
Type: ○ Real ● Virtual
☑ Host Volume ⊐ [   ] ⬆
Label [   ]
Apply   Reset   Close

a) Select the Virtual option to the right of **Type**.

> *You must use* Virtual *because the vertex to be used on the car body is a virtual vertex.*

b) Zoom in to the front of the sedan, so that you can see the front bumper and the first vertex created on the edge at the bottom of the symmetry plane, as shown in Figure 5-19.

c) *Shift*-left-click the first vertex created on the bottom edge of the symmetry plane.

d) *Shift*-left-click the vertex on the sedan that is also on the symmetry plane, as shown in Figure 5-19.

> !   *Make sure that you select the vertex that is on the symmetry plane as well as the sedan. The vertex will be on an orange edge if it is on both the symmetry plane and the sedan geometry.*

e) Click **Apply** to accept the selected vertices and create an edge, as shown in Figure 5-19.
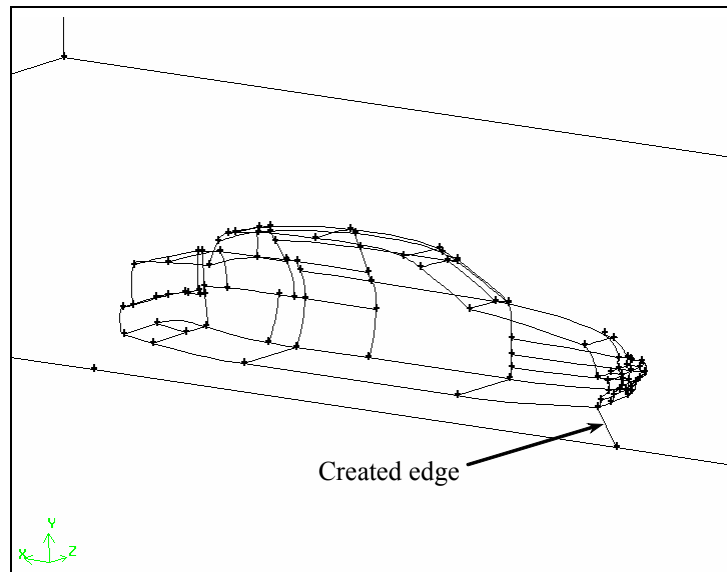
Figure 5-19: Edge from bottom of symmetry plane to front of sedan

3. Create a straight edge from the second vertex created on the bottom edge of the symmetry plane to the rear bumper of the sedan, as shown in Figure 5-20.

   ! *Again, make sure you select the vertex that is on both the sedan geometry and the symmetry plane.*
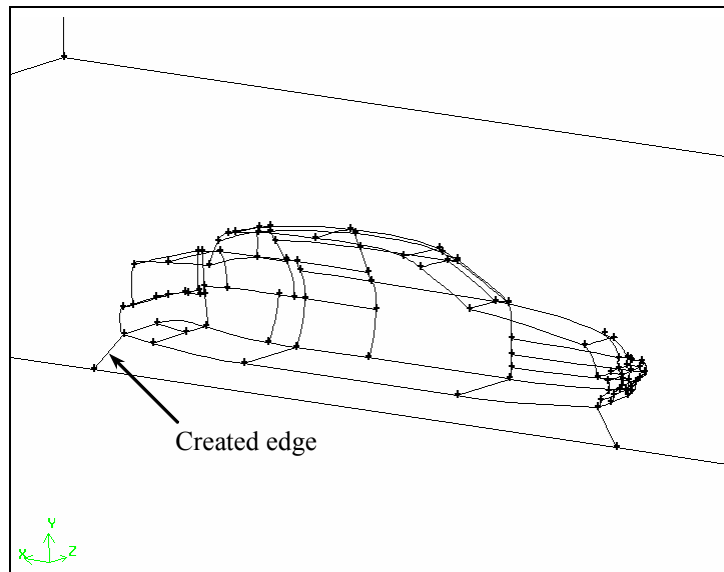
Figure 5-20: Edge from bottom of symmetry plane to rear of sedan

Step 11: Create Faces on the Symmetry Plane

In this step, you will create two new faces on the symmetry plane by stitching edges together. You will use the existing symmetry plane on the brick as a host. The two faces you create in this step will be used to create a volume in the next step.

1. Create a new face on the symmetry plane by stitching edges together.

GEOMETRY 🔲 → **FACE** 🔲 → **FORM FACE** 🔲

This command sequence opens the **Create Face From Wireframe** *form.*

```
┌─────────────────────────────────────┐
│      Create Face From Wireframe     │
├─────────────────────────────────────┤
│                                     │
│  Edges │v_edge.832          │   ▲   │
│                                     │
│  Type:   ◯ Real   ● Virtual         │
│     ☑ Host    Face  ⊐ │face.147│ ▲  │
│     ☐ Hide host                     │
│     Tolerance │0.001│               │
│                                     │
│  Label     │                    │   │
│  ┌─────────┐ ┌─────────┐ ┌────────┐ │
│  │  Apply  │ │  Reset  │ │ Close  │ │
│  └─────────┘ └─────────┘ └────────┘ │
└─────────────────────────────────────┘
```

a) Select the **Type:**Virtual option.

You must use Virtual *because the edges to be selected on the car body are virtual edges.*

b) *Shift*-left-click the four edges underneath the sedan, the two small diagonal edges on the symmetry plane, and the middle edge at the bottom of the symmetry plane.

! *The area under the sedan where the edges to be selected are located is shown in Figure 5-21, and the edges to be selected are shown in Figure 5-22.*
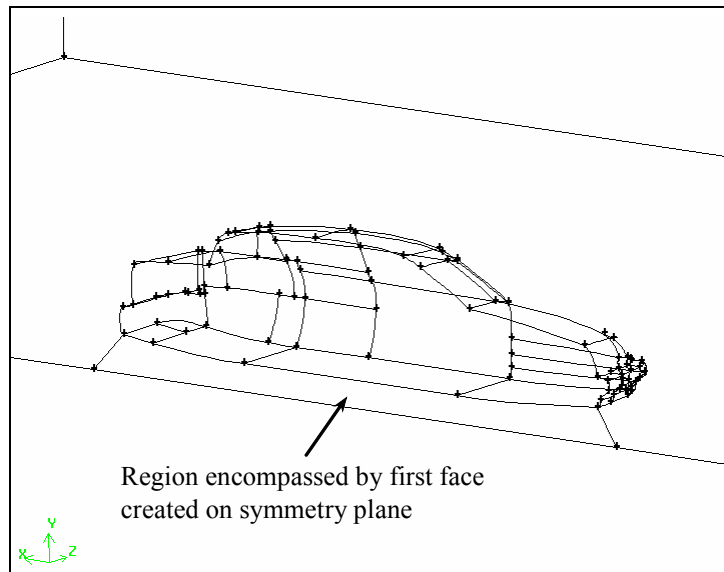
Figure 5-21: Area under sedan where edges to be selected are located

! *You should select seven edges in total. Pay particular attention to any very small edges. If you select an incorrect edge, Shift-middle-click on the edge to deselect it and select the edge next to it.*
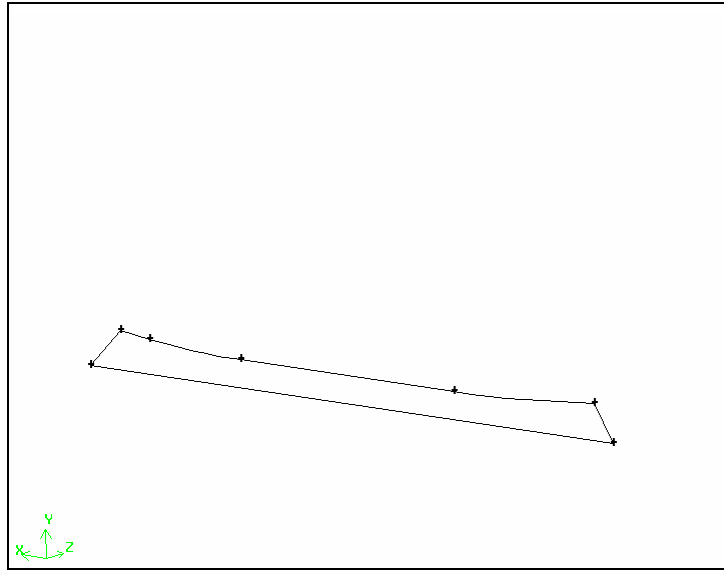
Figure 5-22: Edges used to create face at bottom of sedan

c) Select the Host check box in the **Create Face From Wireframe** form.

d) Select Face from the Host option menu.

e) *Shift*-left-click the back face of the brick (the symmetry plane) in the graphics window, as shown in Figure 5-23.

    If you select the wrong face, Shift-middle-click on the face to deselect it and select the face next to it.

f) Enter 0.001 in the Tolerance text entry box.

g) Click **Apply** to accept the selection and create the face.
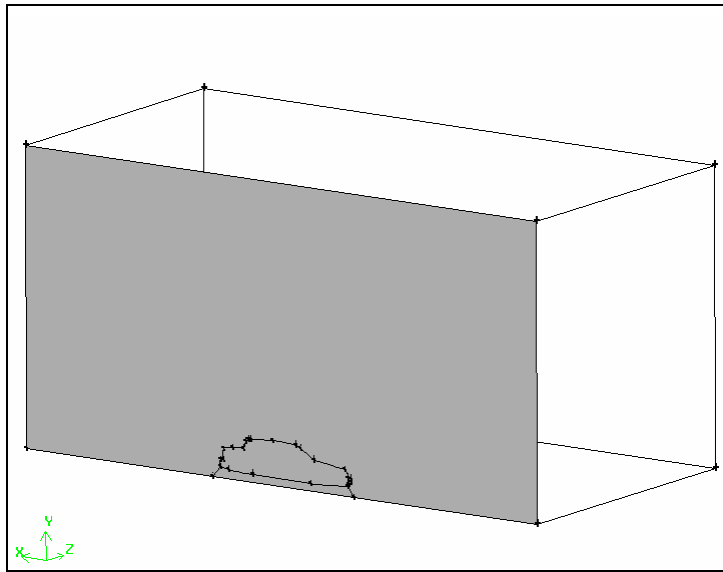
Figure 5-23: Symmetry plane of the brick

2. Create a second face on the symmetry plane.

   a) Check that the Virtual option is selected next to **Type**.

   b) Left-click in the **Edges** list box in the **Create Face From Wireframe** form.

   c) Select all the edges shown in Figure 5-24.

      ! *You should select 25 edges in total.*

   d) Left-click in the list box to the right of Host in the form.

   e) *Shift*-left-click the back face of the brick (the symmetry plane) in the graphics window, as shown in Figure 5-23.

   f) Click **Apply** to accept the selection and create the face.
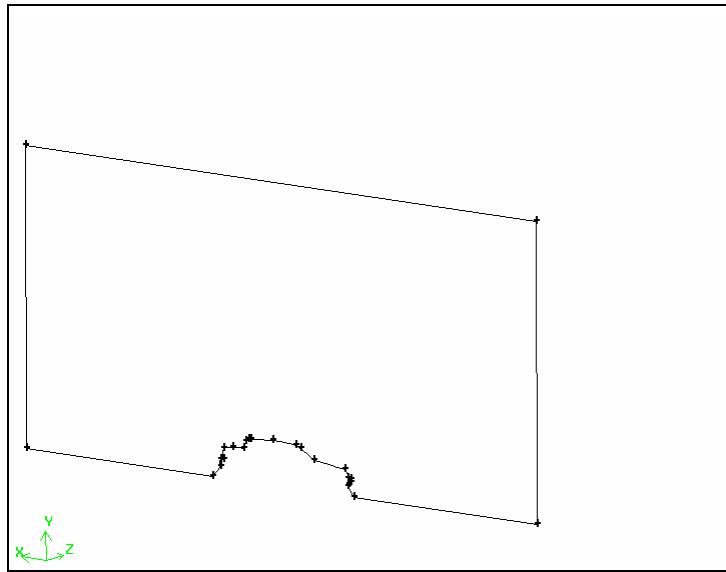
Figure 5-24: Edges used to create face at top of sedan

3. Verify the creation of the faces.

GEOMETRY [icon] → **FACE** [icon] → **SUMMARIZE/QUERY FACES/TOTAL ENTITIES** [icon]

This command sequence opens the **Summarize Faces** *form.*
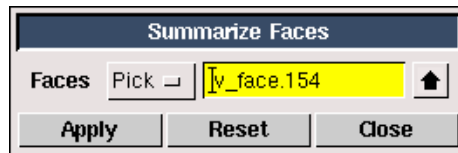
a) Left-click the black arrow to the right of the **Faces** list box.

This action opens the **Face List** *form. There are two types of pick-list forms:* **Single** *and* **Multiple***. In a* **Single** *pick-list form, only one entity can be selected at a time. In a* **Multiple** *pick-list form, you can select multiple entities.*
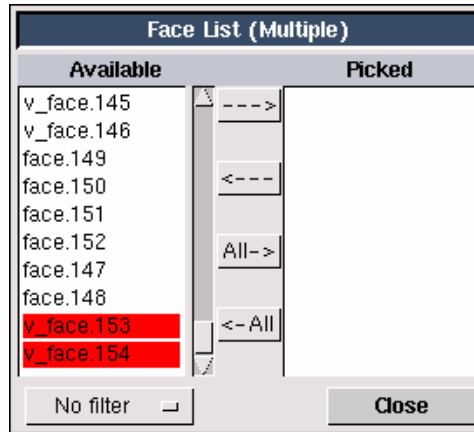
i.   Select the two faces at the bottom of the **Available** list in the **Face List** form.

!   *Note that the names of entities in the* **Available** *list may be different in your geometry. In the form shown above, the last two faces in the* **Available** *list are v_face.153 and v_face.154, but you might see faces with different numbers.*

ii. Click the – –> button to pick the two faces.

The two faces will be moved from the **Available** *list to the* **Picked** *list, and they will be highlighted in the graphics window.*

iii. Check that the two faces highlighted in the graphics window are the correct faces that you should have created in the previous steps.

Figure 5-22 and Figure 5-24 show the faces that you should have created.

iv. Close the **Face List** form.

b) Click **Reset** in the **Summarize Faces** form to unselect the two faces in the graphics window.

Step 12: Create a Volume

1. Use the faces to create a volume.

GEOMETRY 🔲 → **VOLUME** 🔲 → **FORM VOLUME** 🔲

This command sequence opens the **Stitch Faces** *form.*

```
┌─────────────────────────────────────┐
│            Stitch Faces             │
├─────────────────────────────────────┤
│ Faces  face.147                  ↑  │
├─────────────────────────────────────┤
│ Number:   ● Single volume           │
│           ○ Multiple volumes        │
├─────────────────────────────────────┤
│ Type:     ○ Real                    │
│           ● Virtual                 │
│           ○ Real and Virtual        │
│                                     │
│ Tolerance   Auto  ⌐                 │
├─────────────────────────────────────┤
│ Label                               │
├─────────────────────────────────────┤
│   Apply      Reset      Close       │
└─────────────────────────────────────┘
```

a) Select the **Number:**Single volume option.

b) Select the **Type:**Virtual option.

c) Select the symmetry-plane face in the graphics window (as shown in Figure 5-23) and remember the label name (for example, *face.147*).
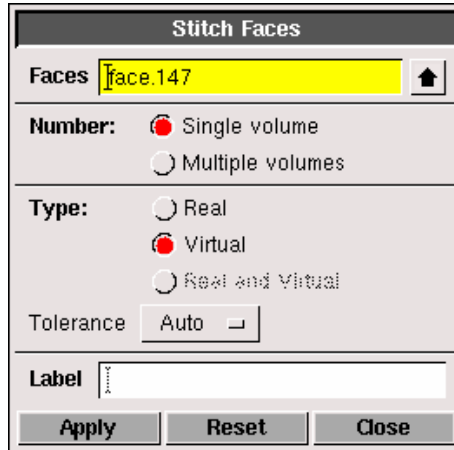
d) Left-click the black arrow to the right of the **Faces** list box.

   This action opens the **Face List** *form.*
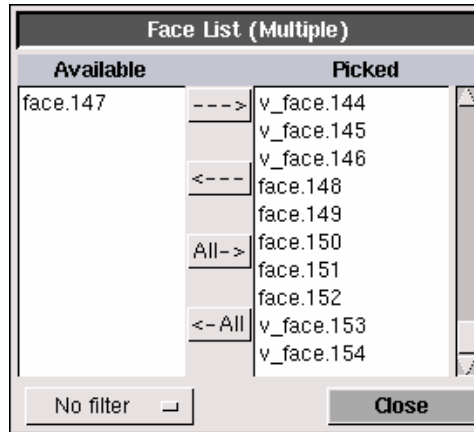
i.   Click on the All –> button to move all the faces from the **Available** list to the **Picked** list.

ii. Select the name of the symmetry-plane face in the **Picked** list.

    The symmetry-plane face will be highlighted in the graphics window.

iii. Click the <––– button to move the symmetry-plane face back into the **Available** list.

iv. Close the **Face List** form.

e) Click **Apply** in the **Stitch Faces** form to accept the selection of the faces and create the volume.

    During the volume creation operation, GAMBIT *redisplays the sedan face meshes. It is advisable, however, to turn off the face mesh displays, so that they do not interfere with the visibility of the entities involved in subsequent steps. To do so, click the* **SPECIFY DISPLAY ATTRIBUTES** *command button*

     at the bottom of the **Global Control** *toolpad, select the* **Mesh:**Off *option on the* **Specify Display Attributes** *form (see Step 7.2, above), and click* **Apply**.
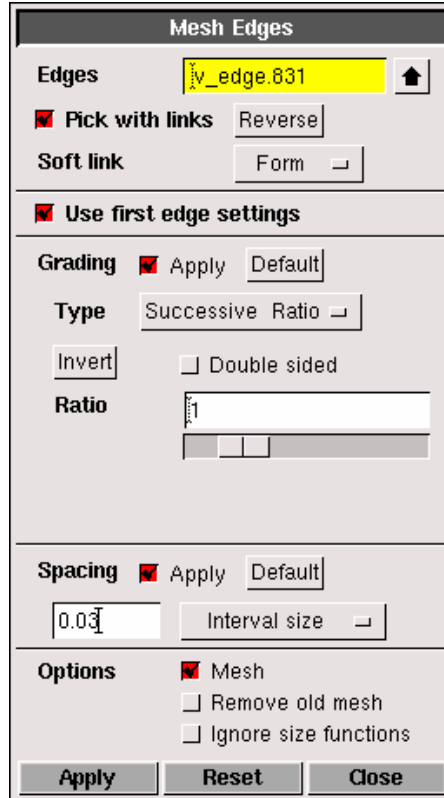
Step 13: Mesh the Edges

When you created the mesh on the faces of the sedan, you used a fine mesh. For the volume, you will create a more coarse mesh, so you will need to instruct GAMBIT to gradually change the mesh density between the coarse and fine meshes. To do this, you will specify the distribution of nodes along some edges in the geometry.

1. Define the grid density on three edges of the geometry underneath the sedan.

MESH [⊞] → **EDGE** [▢] → **MESH EDGES** [✎]

This command sequence opens the **Mesh Edges** *form.*

Mesh Edges
Edges   v_edge.831   ⬆
☑ **Pick with links**   Reverse
Soft link   Form ⊐
☑ **Use first edge settings**
Grading ☑ Apply   Default
Type   Successive Ratio ⊐
Invert   ☐ Double sided
Ratio   1
Spacing ☑ Apply   Default
0.03   Interval size ⊐
Options   ☑ Mesh
☐ Remove old mesh
☐ Ignore size functions
Apply   Reset   Close

a) Select the edges marked A, B, and C in Figure 5-25 (the two small edges you created underneath the sedan and the middle section of the edge underneath the sedan that you split into three sections).

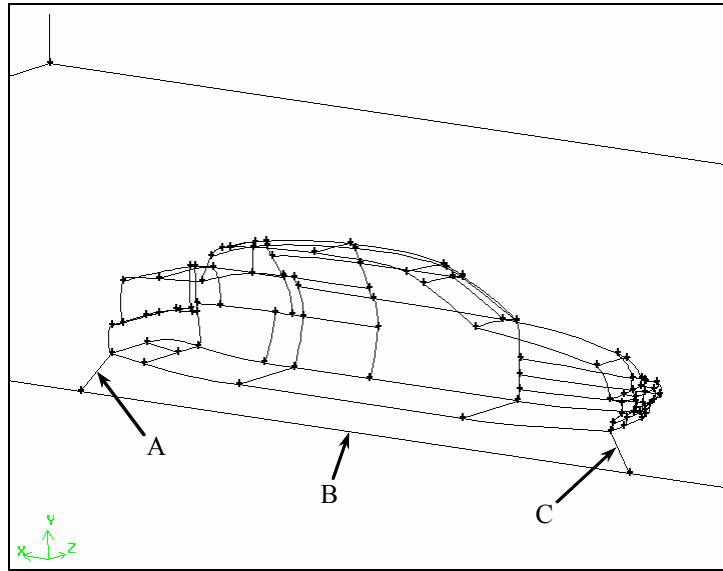*The edges will change color and an arrow and several circles will appear on each edge.*

Figure 5-25: Edges underneath the sedan to be selected for edge meshing

b) Check that **Apply** is selected to the right of **Grading** in the **Mesh Edges** form and that Successive Ratio is selected from the **Type** option menu.

The Successive Ratio *option sets the ratio of distances between consecutive points on the edge equal to the* **Ratio** *specified in the* **Mesh Edges** *form.*

c) Retain the default **Ratio** of 1.

d) Check that **Apply** is selected to the right of **Spacing**. Select Interval size from the option menu under **Spacing** and enter a value of 0.03 in the text entry box.

e) Click the **Apply** button at the bottom of the form.

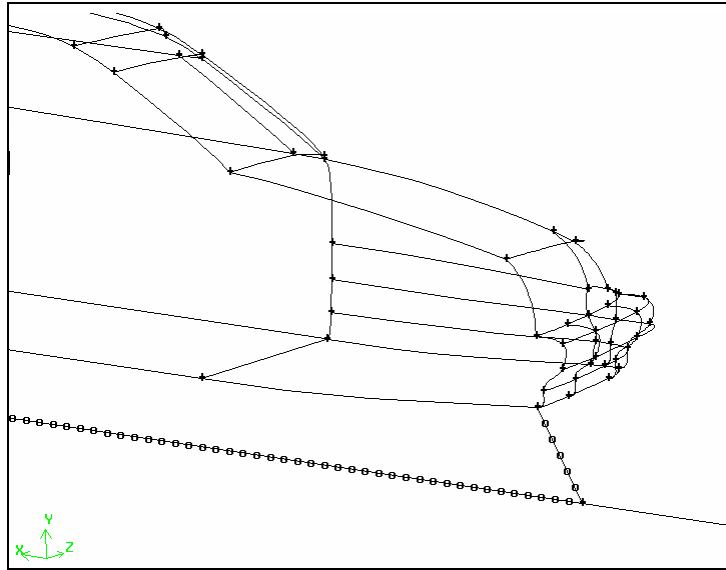*Figure 5-26 shows the mesh on two of the edges underneath the sedan.*

Figure 5-26: Edge meshing near the front of the sedan

2. Set the default variable that invokes flexible grading of the edges.

    a) Select **Defaults** from the **Edit** menu on the main menu bar.

    b) On the **Edit Defaults** form, **Modify** the MESH.EDGE.FLEXIBLE_GRADING default
        variable to set it to 1.

        The flexible-grading default variable controls the manner in which GAMBIT
        *meshes any ungraded edges that are connected to meshed or graded edges. If
        you set the* MESH.EDGE.FLEXIBLE_GRADING *default variable to 1,*
        GAMBIT *grades the edge such that its interval lengths adjacent to the con-
        necting vertex are similar to those on the already meshed or graded edge(s) to
        which it is connected. If an ungraded edge is connected to more than one
        graded or meshed edges at a single vertex, GAMBIT averages the lengths on
        the graded and/or meshed edges to determine the appropriate interval length
        on the ungraded edge.*

    c) **Close** the **Edit Defaults** form.

5-45

Step 14: Mesh the Volume

1. Mesh the volume with a coarser mesh than the mesh on the car faces.

    MESH [⊞] → **VOLUME** [▢] → **MESH VOLUMES** [▨]

    This command sequence opens the **Mesh Volumes** *form.*

Mesh Volumes		
Volumes	v_volume.1	▲
Scheme:	☑ Apply Default	
Elements:	Tet/Hybrid ⊒	
Type:	TGrid ⊒	
☐ **Meshed S.F. on B.L. cap**		
Growth rate	1.2	
Max. size	2	
Spacing:	☑ Apply Default	
1	Interval size ⊒	
Options:	☑ Mesh	
	☐ Remove old mesh	
	☐ Remove lower mesh	
	☐ Ignore size functions	
Apply	**Reset**	**Close**

a) Select the volume in the graphics window.

b) Select Tet/Hybrid from the **Elements** option menu under **Scheme** in the **Mesh Volumes** form, and select TGrid from the **Type** option menu.

    See the GAMBIT *Modeling Guide for more information on meshing schemes.*

c) Retain the default Interval size of 1 under **Spacing** and click the **Apply** button at the bottom of the form.

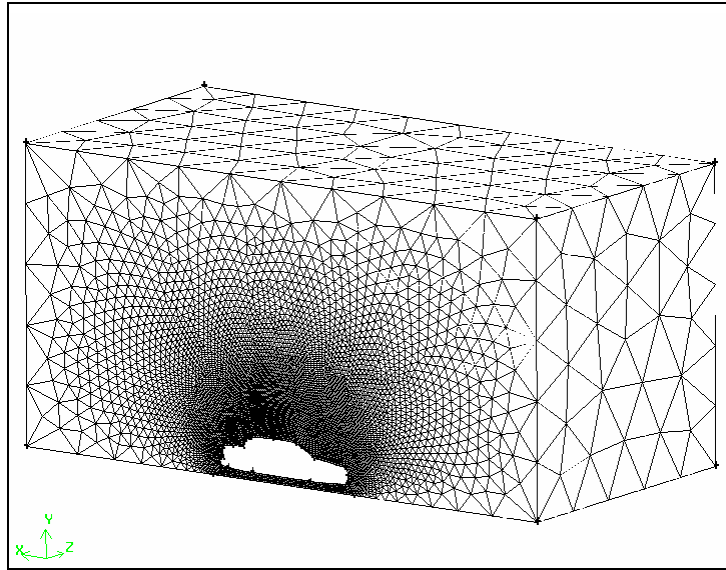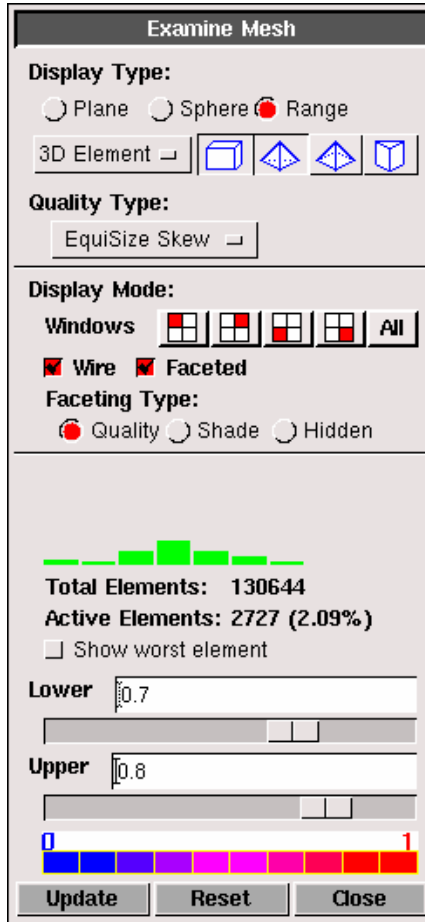   *The surface of the volume mesh is shown in Figure 5-27.*

Figure 5-27: A portion of the volume mesh

Step 15: Examine the Volume Mesh

1. Select the **EXAMINE MESH** command button at the bottom right of the **Global Control** toolpad.

   This action opens the **Examine Mesh** *form.*

   a) Click **Update** at the bottom of the **Examine Mesh** form.

GAMBIT *does* not *automatically update the graphics display when you open the* **Examine Mesh** *form or modify its specifications, such as* **Display Type** *or* **Quality Type**. *To update the graphics display, you must click the* **Update** *pushbutton located at the bottom of the form.* GAMBIT *displays the* **Update** *pushbutton label in red lettering whenever the display needs to be updated to reflect the current* **Examine Mesh** *specifications.*

Some **Examine Mesh** *operations automatically update the graphics display. For example, if you select the* **Display Type:**Range *option and click one of the histogram bars (see below),* GAMBIT *automatically updates the display.*

b) Select Range under **Display Type** at the top of the form.

The 3D Element *type selected by default at the top of the form is a brick* ⬜. *You will not see any mesh elements in the graphics window when you first open the* **Examine Mesh** *form, because there are no hexahedral elements in the mesh.*

c) Left-click on the tetrahedron icon ◇ next to 3D Element near the top of the form.

The mesh elements will now be visible in the graphics window.

d) Select EquiSize Skew from the **Quality Type** option menu.

This is the default skewness measure for tetrahedra in TGrid.

e) Left-click the histogram bars that appear at the bottom of the **Examine Mesh** form to highlight elements in a particular quality range.

Figure 5-28 shows the view in the graphics window if you click on the fifth bar from the right on the histogram (representing cells with a skewness value between 0.7 and 0.8). These low values for the maximum skewness indicate that the mesh is acceptable.

The histogram consists of a bar chart representing the statistical distribution of mesh elements with respect to the specified **Quality Type**. *Each vertical bar on the histogram corresponds to a unique set of upper and lower quality limits.*
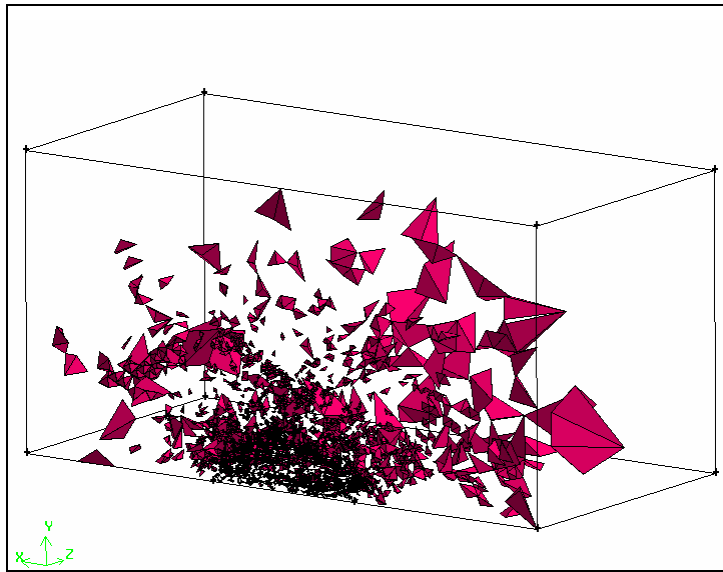
Figure 5-28: Elements within a specified quality range

f) Close the **Examine Mesh** form by clicking the **Close** button at the bottom of the form.

Step 16: Set Boundary Types

1. Remove the mesh from the display before you set the boundary types.

   This makes it easier to see the edges and faces of the geometry. The mesh is not deleted, just removed from the graphics window.

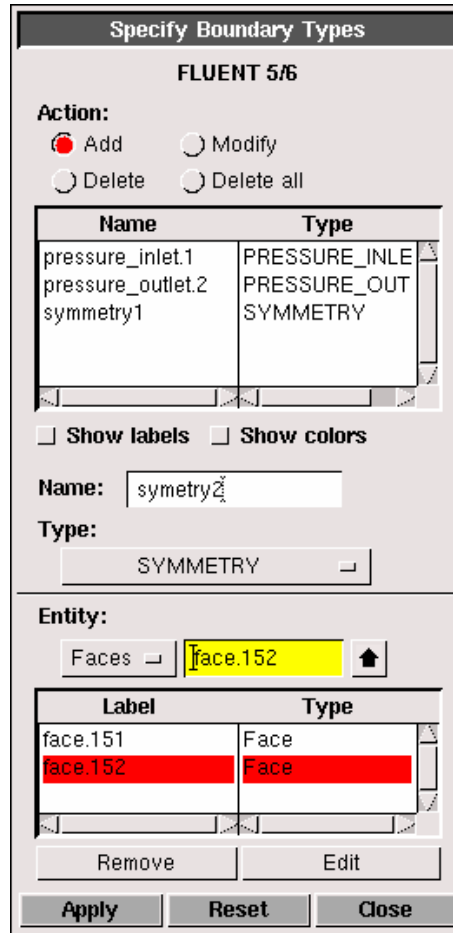   a) Click the **SPECIFY DISPLAY ATTRIBUTES** command button at the bottom of the **Global Control** toolpad.

   b) Select the Off radio button to the right of **Mesh** near the bottom of the form.

   c) Click **Apply** and close the form.

2. Set boundary types for the sedan.

   ZONES → **SPECIFY BOUNDARY TYPES**

   *This command sequence opens the **Specify Boundary Types** form.*

a) Define the pressure inlet boundary.

   i.   Select PRESSURE_INLET in the **Type** option menu.

   ii. Check that Faces is selected as the **Entity**.

   iii. *Shift*-left-click the face on the brick in front of the car in the graphics window
        (marked A in Figure 5-29) and click **Apply** to accept the selection.

        This face will be set as a pressure inlet.

GAMBIT *will give the boundary a default name based on what you select in the* **Type** *and* **Entity** *lists (pressure_inlet.1 in this example). You can also specify a name for a boundary by entering a name in the* **Name** *text entry box.*
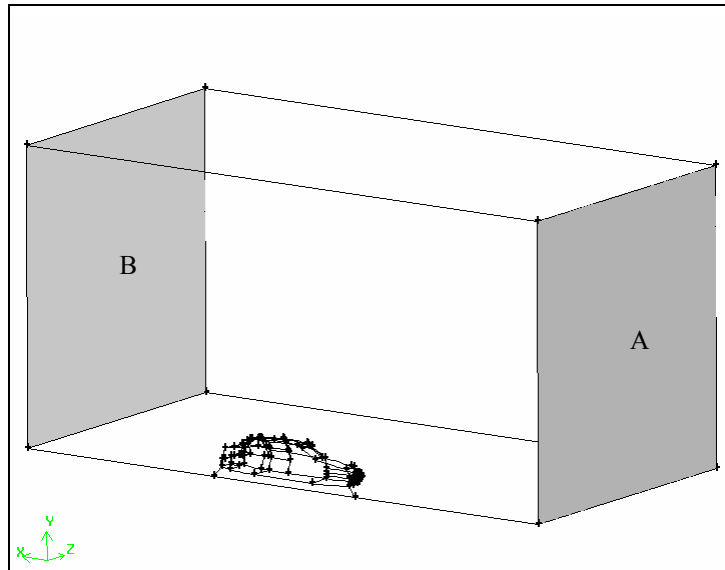

Figure 5-29: Pressure inlet (A) and pressure outlet (B) for the sedan geometry

b) Define the pressure outlet boundary.

   i.   Change the **Type** to PRESSURE_OUTLET by selecting it from the option menu below **Type**.

   ii. Select the face on the brick behind the car in the graphics window (marked B in Figure 5-29) and click **Apply** to accept the selection.

c) Define symmetry boundary types for the two faces on the symmetry plane of the brick.

   i.   Enter "symmetry1" in the **Name** text entry box.

   ii. Select SYMMETRY from the **Type** option menu.

   iii. Select the two faces you created on the symmetry plane of the brick (the faces marked C and D in Figure 5-30) and click **Apply** to accept the selection.

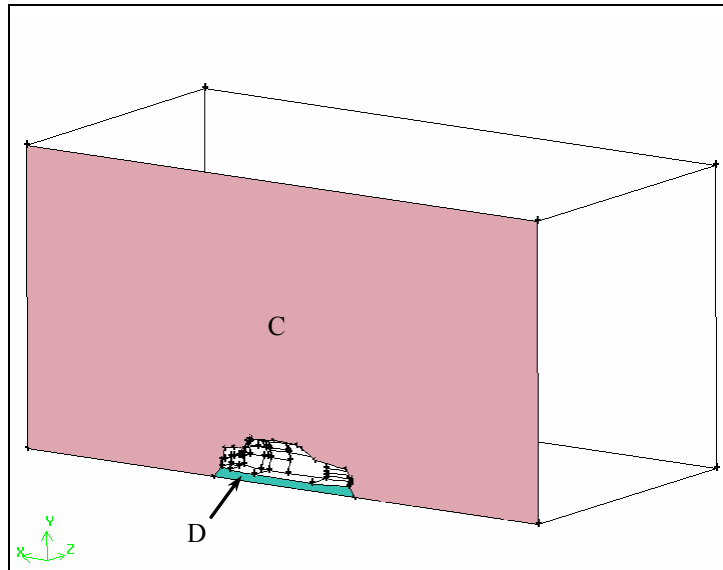   GAMBIT *will merge the two faces into a single symmetry zone.*

Figure 5-30: Two faces created on the symmetry plane of the brick

d) Define symmetry boundary types for the top face of the brick and the side face opposite the symmetry plane.

   i.   Enter "symmetry2" in the **Name** text entry box.

   ii. Check that SYMMETRY is selected in the **Type** option menu.

   iii. Select the faces on the brick that are above and to the side of the sedan (the faces marked E and F in Figure 5-31) and accept the selection.
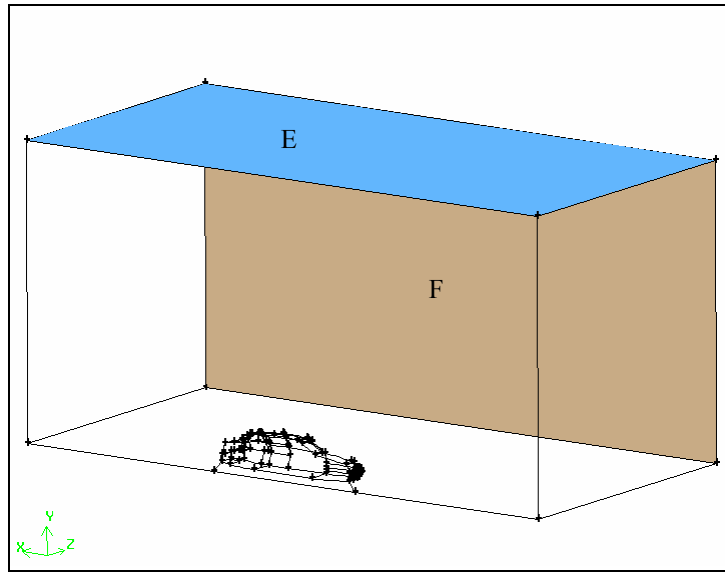
Figure 5-31: Two symmetry boundaries for the sedan geometry

The pressure inlet, pressure outlet, and symmetry boundaries for the sedan geometry are shown in Figure 5-32. (<u>NOTE</u>: To display the boundary types in the graphics window, select the **Show labels** *options on the* **Specify Boundary Types** *form. To display colors associated with each boundary-type assignment, select the* **Show colors** *option.)*
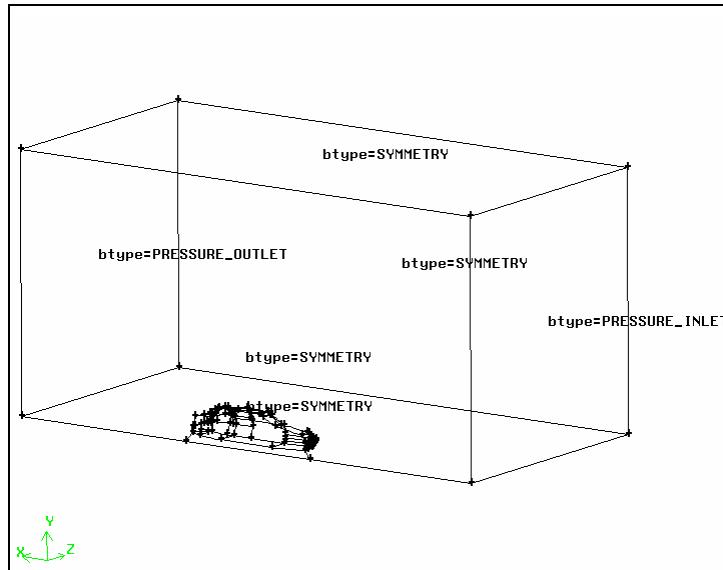
Figure 5-32: Boundary types for the sedan geometry

Note that you could also specify the remaining outer edges of the sedan geometry as wall boundaries. This is not necessary, however, because when GAMBIT *saves a mesh, any faces (in 3D) on which you have not specified a boundary type will be written out as wall boundaries by default.*

In addition, when GAMBIT *writes a mesh, any volumes (in 3D) on which you have not specified a continuum type will be written as fluid by default. This means that you do not need to specify a continuum type in the* **Specify Continuum Types** *form for this tutorial.*

Step 17: Export the Mesh and Save the Session

1. Export a mesh file for the sedan.

   File → Export → Mesh...

   This command sequence opens the **Export Mesh File** *form.*
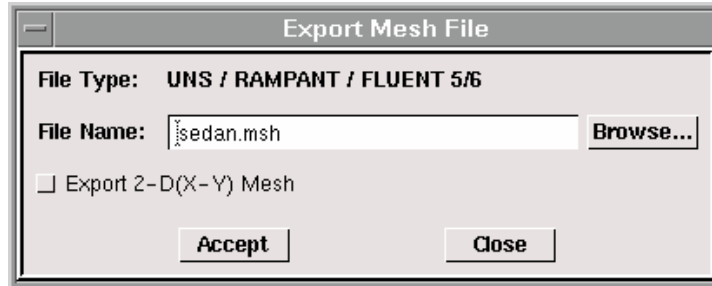
   a) Enter the **File Name** for the file to be exported (sedan.msh).

   b) Click **Accept** in the **Export Mesh File** form.

      The file is written to your working directory.

      (NOTE: During mesh export, GAMBIT *displays a warning message in the* **Transcript** *window indicating that at least one geometric entity in the model is unmeshed. The unmeshed entity is the real host symmetry-plane face (see Step 11), which is not included as part of the flow volume (see Step 12).)*

2. Save the GAMBIT session and exit GAMBIT.

   File → Exit

   GAMBIT *will ask you whether you wish to save the current session before you exit.*
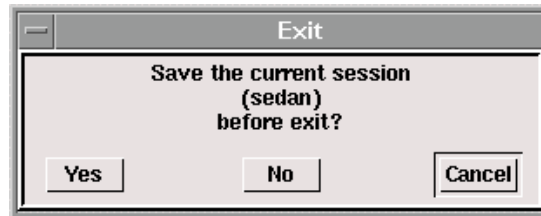
   a) Click **Yes** to save the current session and exit GAMBIT.

5.5 Summary

This tutorial illustrated how to import geometry from an external CAD package as an IGES file, and mesh it. Several geometry "cleanup" operations were demonstrated. Additional geometry was created to construct a box around the car-body geometry, and an unstructured tetrahedral volume mesh was generated.