GAMBIT TUTORIAL 03
SIMULATION OF WIND AROUND BUILDINGS

GAMBIT TUTORIAL 03: SIMULATION OF WIND AROUND BUILDINGS

Purpose

Simulation of wind conditions around buildings is important due to the need to analyze pedestrian wind comfort and wind resources around buildings. For high rise buildings many urban authorities require wind comfort studies (by wind tunnel or CFD) before a building permit is granted.
The purpose of this tutorial is to provide guidelines and recommendations for creating and meshing a CFD model for simulation of wind around buildings. The first part of the tutorial teaches you how to create geometry of the model while the second part gives you step by step guidance to mesh the model. In the second part of the tutorial, hexahedron mesh is used to mesh the model. Generally speaking a hexahedron mesh has a better mesh quality and a high spatial efficiency than a tetrahedron mesh, but it is more time consuming and more difficult to generate hexahedron mesh. For most cases, geometry decomposition is needed in order to generate a hexahedron mesh.

Prerequisites

This tutorial assumes that you are familiar with the Gambit interface and that you have gone through the introductory tutorials on geometry creation and mesh generation. In this tutorial, you will create the model of the Canal Buildings and the neighborhood around in Gladsaxe. As shown in Fig. 1, there are 22 buildings with a building height of max. 50 m. For the sake of simplicity, small buildings, trees and the bushes are not considered in the model. The 12 buildings to be included in the model are given in Fig. 2.

Fig. 1 Canal Buildings in Gladsaxe.

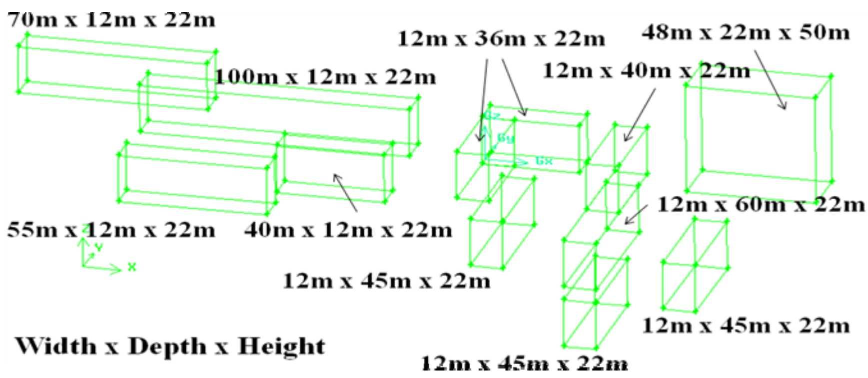


Fig. 2 Buildings to be considered

As the modelling goal is not on the buildings themselves but on the air flow around the buildings, the space around the buildings has to be considered. In order to minimize the channel effect, the space around the building (domain) shall be large enough. The following three requirements are recommended for the size of the domain.

(1), There should be sufficient distances between the buildings and the domain boundaries. The rule of thumb of the domain size is given in Fig. 3. If the tallest building has a height of H_{max}, the height of the domain has to be larger than $6H_{max}$. The distance between the buildings and the side boundaries and the upwind boundary has to be larger than 5 H_{max} while there should be a distance of 15 H_{max} between the buildings and the leeward boundary.

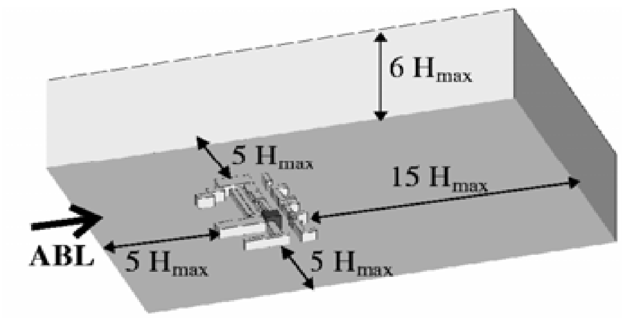


Fig. 3 Minimal distances between the buildings and the domain boundaries. (H_{max}: the maximum height of the buildings. ABL: Atmospheric Boundary Layer, indicating wind direction.)

(2), The blockage ratio of any cross section of the domain shall be no larger than 3-5%.

The blockage area in a cross section of the domain is illustrated in Fig. 4 where A_{domain} and $A_{building}$ is the cross section area of the domain and the buildings respectively. The blockage ratio, BR, shall be:

$$BR = \frac{A_{building}}{A_{domain}} < 3 - 5\%$$

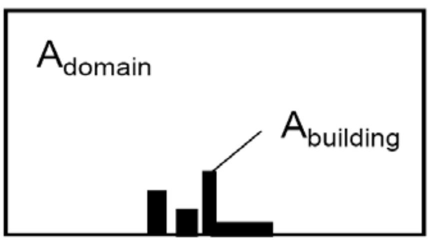
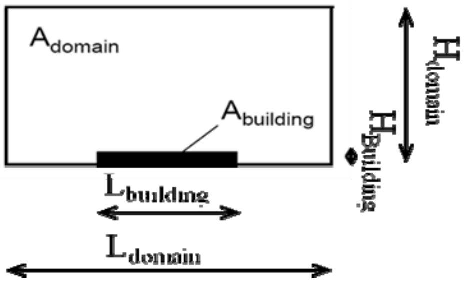


Fig. 4 Blockage area in a cross section of the domain.

(3), Directional blockage ratio shall be smaller than 17-22%.



$$BR_L = \frac{L_{building}}{L_{domain}} \ \& \ BR_H = \frac{H_{building}}{H_{domain}} < 17-22\%$$

Fig. 5 Directional blockage ratio.

$L_{building}$ and L_{domain} are the length of the buildings and the domain respectively. $H_{building}$ and H_{domain} are the height of the buildings and the domain respectively.

1 Geometry creation

1.1 Create the buildings

Start Gambit.

Since the aim is to generate a meshed model for Fluent 6.2/6.3, the solver should be specified as Fluent 5/6 by the menu \Sover\Fluent 5/6.\

1.1.1 Create the first building 'blok 1'

The buildings are created using the top down method.

Let's create the first building 'blok 1' using the top down method. As shown in Fig. 6, Gambit can directly create a brick with specification of the dimensions of the building. The brick created is shown in Fig. 7. The coordinate symbol in green shows directions of the X, Y, Z axis.

Do the followings to create the building:

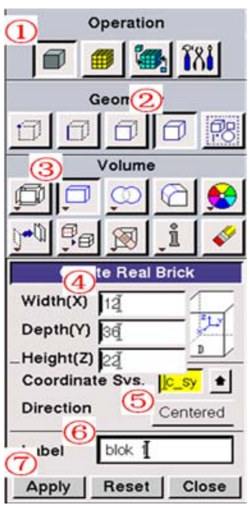


Fig. 6 Creation of a brick

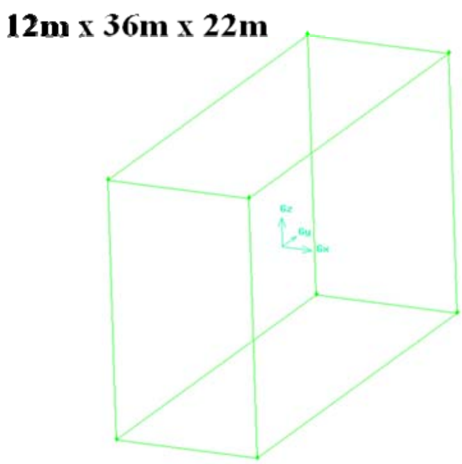


Fig. 7 A building created using top down method

① Click pad 'Geometry'

② Click pad 'Volume'

③ Create real brick

④ Key in width 12, depth 36 and height 22

⑤ Select direction, Centered

⑥ Key in label 'blok 1'

⑦ Apply

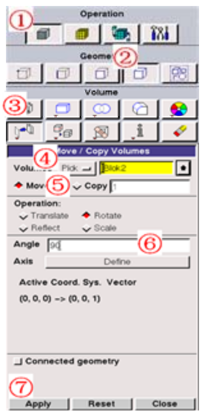
Note: *⑤, Using Centered as the direction will create a building with the origin of the coordinate system (0,0,0) located in the center of the building. Using +X+Y+Z as the direction will create a building with the origin of the coordinate system (0,0,0) located in a corner of the building.

*⑥, It is not necessary to key in a label. If not specified, a label will be automatically given by the program, which is normally ‘volume. xxx’. (‘xxx’ is a serial number.)

1.1.2 Create the second building ‘blok 2’ and move it to the right place

(1) The building ‘blok 2’ has the same size as the building ‘blok 1’. Create the building ‘blok 2’ by repeating the step of chapter 1.1.1 with the same settings except a label of ‘blok 2’.

(2) Then rotate the building 90˚ around +Z axis as shown in Fig. 8. The buildings ‘blok1’ and ‘blok2’ are shown in Fig. 9 after rotation. (‘blok2’ is in red.)



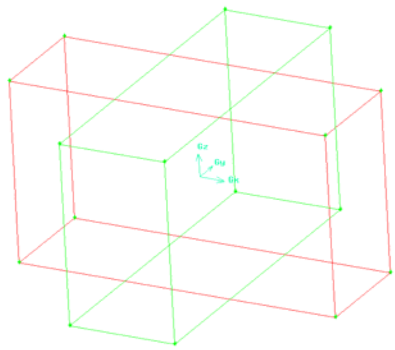


Fig. 8 Rotate the building ‘blok 2’ along +Z axis.

① Click pad ‘Geometry’

② Click pad ‘Volume’

③ Move/copy volume

④ Select the volume ‘blok 2’

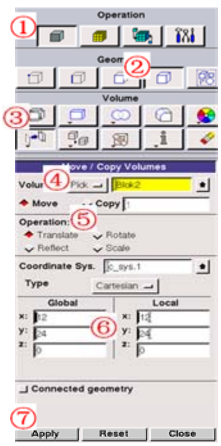
Fig. 9 The two buildings after rotation .

⑤ Select Move and Rotate

⑥ Key in an angle of 90˚

⑦ Apply

(3) Then translate the building ‘blok 2’ to the right place, shown in Fig. 10.



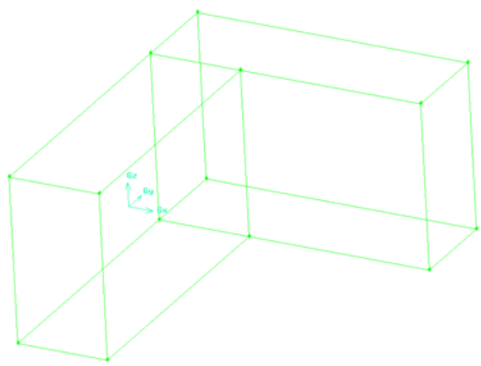


Fig. 10 Translate the building ‘blok 2’.

① Click pad ‘Geometry’

② Click pad ‘Volume’

③ Move/copy volume

④ Select the volume ‘blok 2’

Fig. 11 The two buildings after translation.

⑤ Select Move and Translate

⑥ Key in distances of 12, 24 and 0 in x, y, z direction respectively.

⑦ Apply

1.1.3 Create the other 10 buildings and move them to the right places

Create the building ‘blok3’ with width 12 depth 40 height 22.

Move/Translate the volume "Blok3" with an offset (48, 2, 0).

Create the building ‘volume.4’ with width 48 depth 22 height 50.

Move/translate the volume "volume.4" with an offset (96, 20, 14).

Create the building "BlokAA1" with width 40 depth 12 height 22.

Move/translate the volume "BlokAA1" with an offset (-50, -22, 0).

Create the building "BlokAA2" with width 55 depth 12 height 22.

Move/translate the volume "BlokAA2" with an offset (-95, -45, 0).

Create the building "BlokAA3" with width 100 depth 12 height 22.

Move/translate the volume "BlokAA3" with an offset (-83, 25, 0).

Create the building "BlokAA4" with width 70 depth 12 height 22.

Move/translate the volume "BlokAA4" with an offset (-150, 52, 0).

Create the building "BlokBB1" with width 12 depth 45 height 22.

Move/translate the volume "BlokBB1" with an offset (22, -60, 0).

Create the building "BlokBB2" with width 12 depth 60 height 22.

Move/translate the volume "BlokBB2" with an offset (60, -65, 0).

Create the building "BlokBB3" with width 12 depth 45 height 22.

Move/translate the volume "BlokBB3" with an offset (100, -85, 0).

Create the building "BlokBB4" with width 12 depth 45 height 22.

Move/translate the volume "BlokBB4" with an offset (75, -130, 0).

The final building models are shown in Fig. 12.

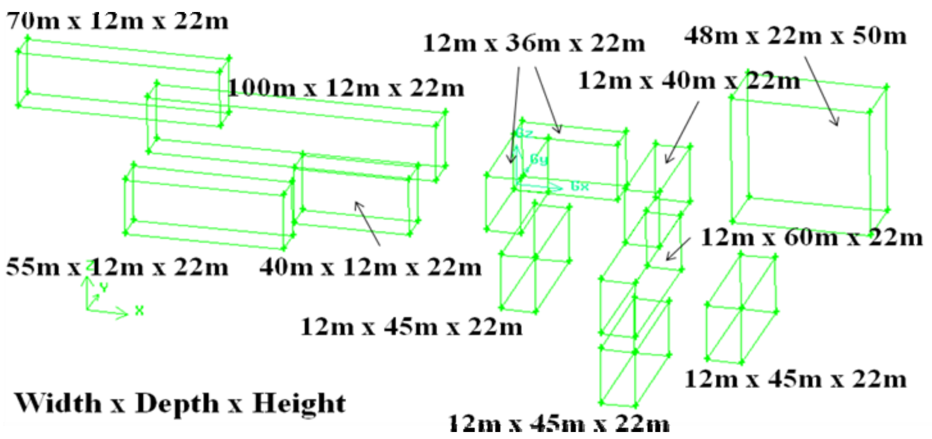


Fig. 12 Final building models.

You shall check the model before going on the next chapter.

1.2 Create a large brick for the domain and create a small brick for mesh refinement

(1) Create a large brick for the domain:

Create a volume "Domain" with width 1000 depth 1200 height 300.

Move/translate the volume "Domain" with an offset (0, 200, 139).

(2) Create a small brick for mesh refinement area:

Create a volume "Domain small" with width 360 depth 260 height 60.

Move/translate the volume "Domain small" with an offset (-30, -40, 19).

The model is shown in Fig. 13. Now the domains (domain and domain small) and the buildings are created but they are not actually connected. In order to connect the buildings with the domains, the buildings and the small domain have to be imprinted (split) into the domain.

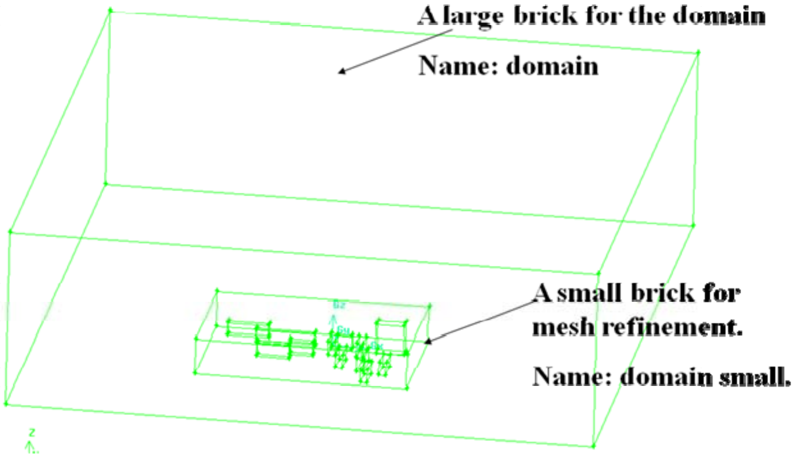


Fig. 13 The model.

1.3 Imprint the buildings and the small domain in the domain

(1) To imprint the buildings and the small domain into the domain, the domain has to be split by the buildings and the small domain, see Fig. 14.

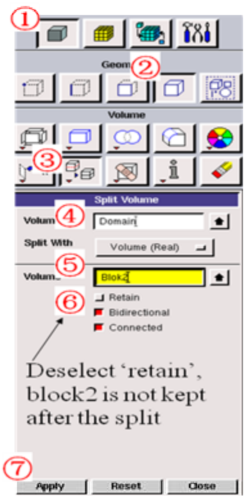


Fig. 14 Split the domain with the building 'block2'.

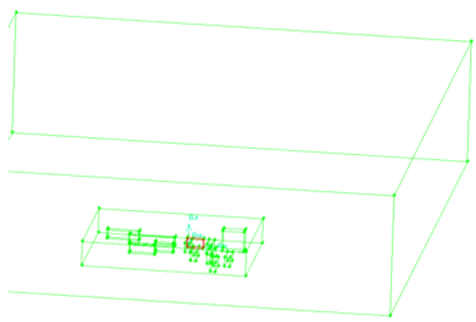


Fig. 15 The building 'blok 2' marked in red.

① Click pad 'Geometry'

② Click pad 'Volume'

③ Split volume

④ Select the volume 'domain'

⑤* Select volume (real) to split with and pick the volume 'blok 2'.

⑥* Deselect 'retain'.

⑦ Apply

Note: *⑤, If the volume is to be split with another volume, select volume (real) and pick up the volume; If the volume is to be split with a face, select face (real) and pick up the face.

*⑥, Deselect 'retain' so that the volume 'domain small' is not kept after the split. Select 'retain' means keeping the volume 'domain small' after the split. Here 'retain' is deselected because there is a duplicate of 'blok2' by split from the domain.

(2) Repeat the steps shown in Fig. 14 to imprint all the rest of the buildings and the small domain into the domain.

1.4 Delete the buildings

Since the aim of the tutorial is to investigate the air flow around buildings, the buildings themselves are not of our interest and therefore shall be deleted.

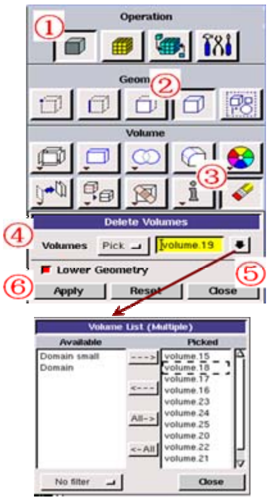


Fig. 16 Split the domain with the building 'block2'.

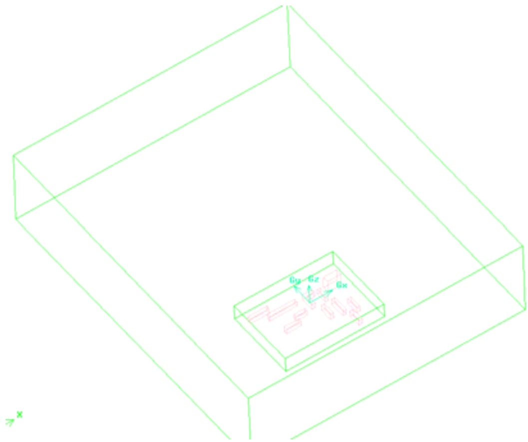


Fig. 17 The building 'blok 2' marked in red.

① Click pad 'Geometry'

② Click pad 'Volume'

③ Delete volume

④ Pick all the buildings

⑤* Select list can be used to pick up volumes.

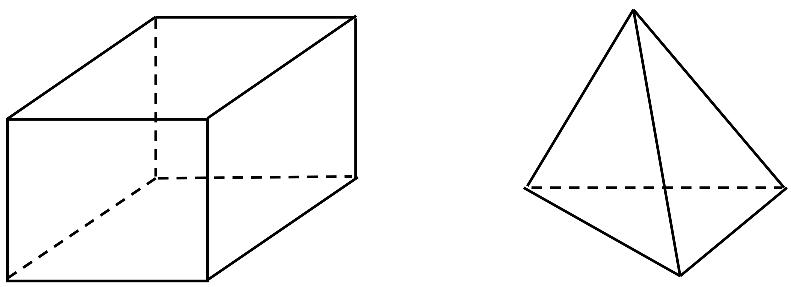
⑥* Enable 'lower geometry' and apply.

Note: *⑤, Can be activated by clicking on the arrow.

*⑥, To enable 'lower geometry' means that all the lower geometry (faces and vertices) will be deleted as well. To disable 'lower geometry' means that lower geometry (faces and vertices) will not be deleted.

2 Mesh generation

In this part of the tutorial, the model will be meshed with a hexahedron mesh. Generally speaking a hexahedron mesh has a better mesh quality and a high spatial efficiency than a tetrahedron mesh (see Fig. 18), but it is more time consuming and more difficult to generate hexahedron mesh. For most cases, geometry decomposition is needed in order to generate a hexahedron mesh. The tutorial gives one example of geometry decomposition.



hexahedron tetrahedron

Fig. 18 Hexahedron and tetrahedron mesh elements.

2.1 Model decomposition

There are no concrete rules for model decomposition. For the same model, there can be many different ways of decomposition. Here are some guidelines for the building models as shown in Fig. 17.

- A denser mesh is needed in the region close to the ground, therefore decomposition is necessary to create the region, see Fig. 19. The height of the region could be slightly (for example, 5 m) higher than the maximal building height, H_{max}. The height of the region is the same as the height of the 'domain small'.
- A denser mesh is needed for the area adjacent to the buildings.
- Decomposition can be done in such a way that the area from the buildings to the top of the domain can be separated.
- Decomposition is needed if there is interior loop hanging in the volume;
- How to decompose: to split the volume with the help of face/faces and volume/volumes.

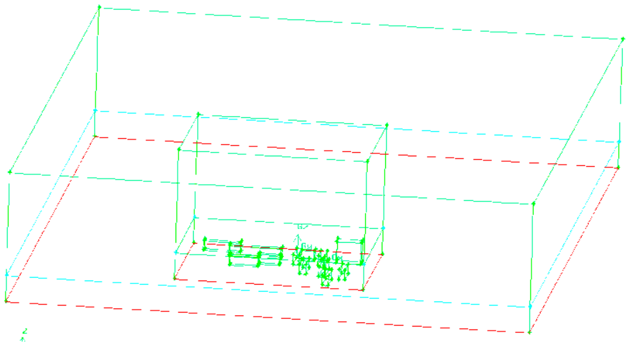


Fig. 19 The building model after decomposition.

Now let do the following steps to decompose the model.

2.1.1 Create a brick 'helpvol1' and split the domain with the brick

(1) Create a help volume 'helpvol1' which can be used as a cutting tool later.

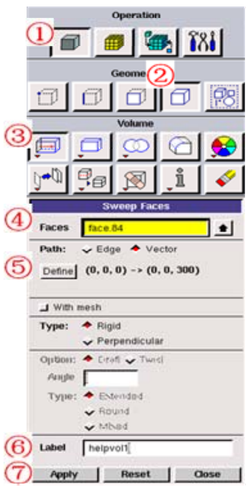


Fig. 20 Create a brick 'helpvol1' to be used as a cutting tool later.

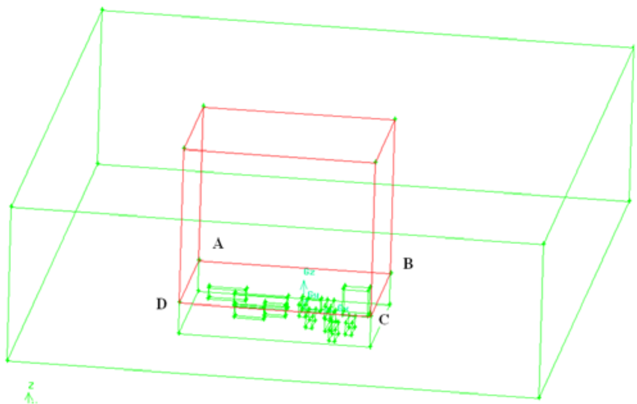


Fig. 21 The brick 'helpvol1' marked in red.

① Click pad 'Geometry'

② Click pad 'Volume'

③* Create volume by Sweep faces

④* Pick the face ABCD (see Fig. 21)

⑤ Define the path vector as (0,0,300).

⑥ Key in label 'helpvol1'.

⑦ Apply

Note: *③, Right button click to activate the list, select sweep faces.

*④, Find the face indicated as ABCD in Fig. 21 and pick the face. The name of the face may not be face. 84 as shown in Fig. 20.

(2) Split the domain with the volume ‘helpvol1’.

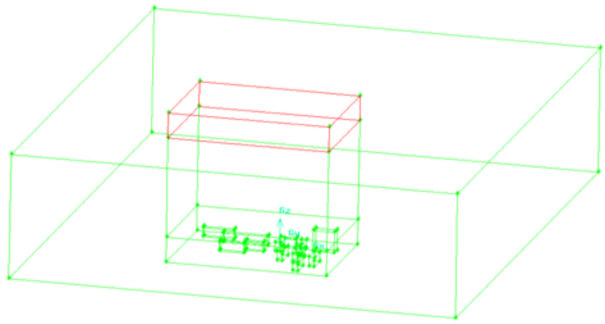


Fig. 22 Split the domain with ‘helpvol1’.

① Click pad ‘Geometry’

② Click pad ‘Volume’

③ Split volume

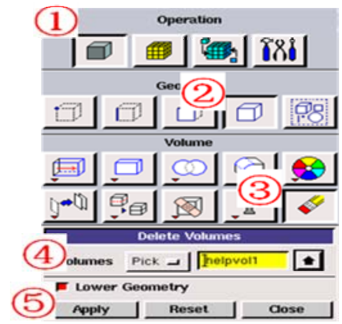
④ Pick the volume ‘domain’.

Fig. 23 The model after the split.

⑤ Select volume (real) to split with and pick the volume ‘helpvol1’.

⑥ Apply.

(3) Delete the volume leftover (marked in red in Fig. 25).



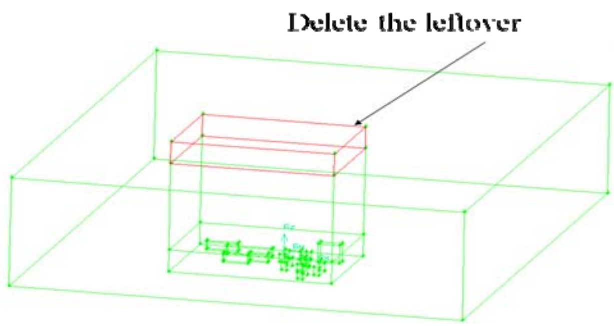


Fig. 24 Delete the leftover.

① Click pad ‘Geometry’

② Click pad ‘Volume’

③ Delete volume

Fig. 25 The brick ‘helpvol1’ marked in red.

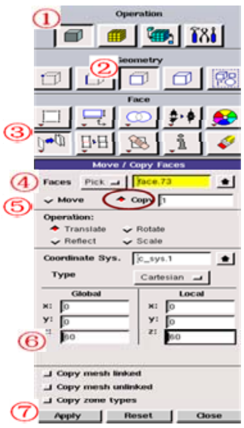
④ Pick the volume ‘helpvol1’.

⑤ Apply.

2.1.2 Create a face and split the domain with the face

It is necessary to assign a denser mesh in the region close to the ground. The region is separated from the domain by Split with a help face. The domain shall be split by the plane of the top surface of the “domain small”. The help face can be created by a copy of the ground face as ABCD shown in Fig. 27.

(1) Create a help face by Copy/Translate.



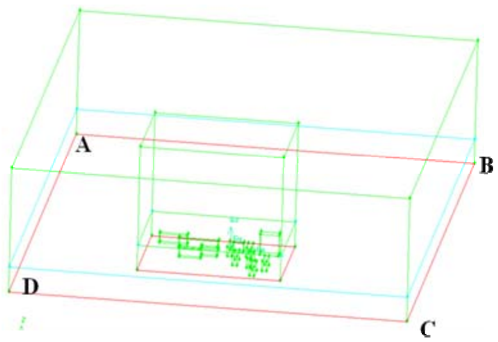


Fig. 26 Create a face by copy

① Click pad 'Geometry'

② Click pad 'Face'

③ Move/Copy faces

④ Pick the face ABCD (see Fig. 27)

Fig. 27 The brick 'helpvol1' marked in red.

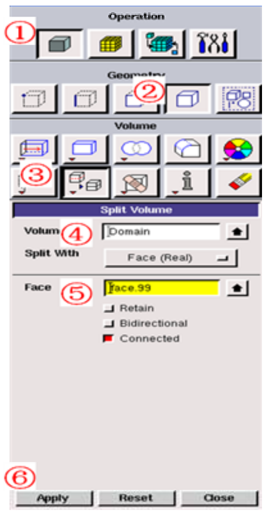
⑤ Select Copy/Translate.

⑥* Key in distance, 60 in Z direction..

⑦ Apply

*⑥, a distance of 60 m in Z direction makes the copy of the face ABCD lie in the plane as the top surface of the 'domain small'.

(2) Split the domain by the help face marked in red in Fig. 29.



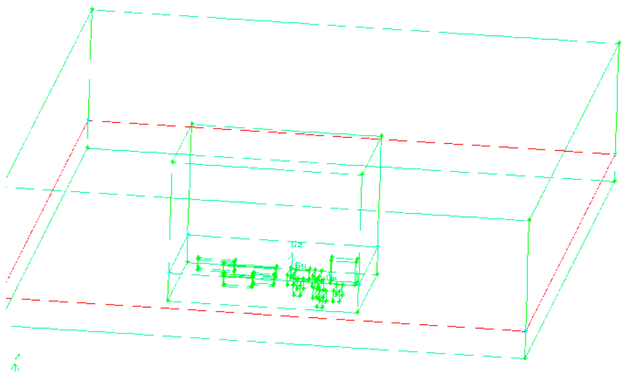


Fig. 28 Split the domain with the face.

① Click pad 'Geometry'

② Click pad 'Volume'

③ Split volume

④ Pick the volume 'domain'.

Fig. 29 The model after the split.

⑤ Select face (real) to split with and pick the face marked in red in Fig. 29.

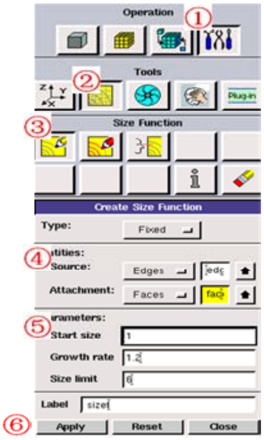
⑥ Apply.

2.2 Mesh the decomposed model

2.2.1 Define size function for the bottom face of the small domain

It is important to have sufficient number of cells in the channels between buildings and along the building height. As a guideline, there should be at least 10 cells in the space between buildings and there shall be at least 10 cells along the building height. In case where the distance between buildings is small, a mesh with uniform density will end up with too few cells in these areas. Size function can be used to assign a denser mesh in the spaces between buildings, thus fulfilling the requirement with few cells.

Define size function for the bottom face of the volume 'domain small' as follows.



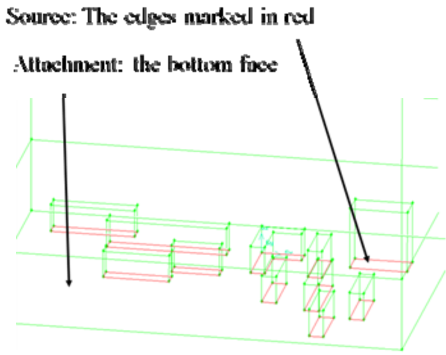


Fig. 30 Define size function for the bottom face of the small domain.

① Click pad 'Tools'

② Click pad 'Size function'

③ Create a size function

④ Select edges marked in red as source; select the bottom face of the small domain as attachment, see Fig.31.

Fig. 31 Size function with the bottom face as attachment and the edges marked in red as source.

⑤* Key in a start size of 1, a growth rate of 1.2 and a size limit of 6.

⑥ Apply.

Note: *⑤, The specifications of start size, growth rate and size limit are case related.

2.2.2 Define boundary layer mesh for the two bottom faces

As a rule of thumb, grid line shall align with the flow. Generally wind flows along the ground if there are no objects such as buildings and bushes, therefore the grid line adjacent to the ground shall be parallel to the ground. A boundary layer mesh can be used to generate a denser mesh with grid lines parallel to the ground. It shall be mentioned that special attention shall be paid to the depth of the first row when assigning boundary layer mesh to the ground surface. The depth of the first row shall be twice the roughness height of the ground.

Do the follows to define the boundary layer mesh:

As shown in Fig.32,

① Click pad 'Mesh'

② Click pad 'Boundary layer'

③ Create boundary layer

④* Key in depth of the first row, 0.8 m.

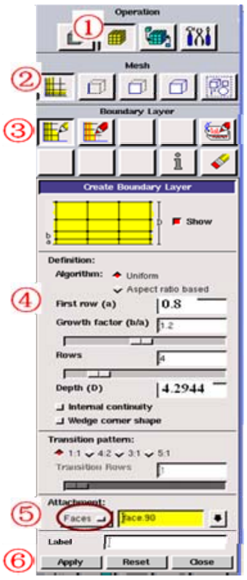
⑤* Select face ABCD and abdc to attach.

⑥ Apply.

*④, The depth of the first row is case sensitive which differs from case to case. **As a rule of thumb, depth of the first row shall be twice the roughness height of the ground.**

*⑤, The two bottom faces are shown in Fig. 33 as ABCD and abcd. Face ABCD + abcd = the ground.

The boundary layer mesh turns white after the command is applied in step ⑥.



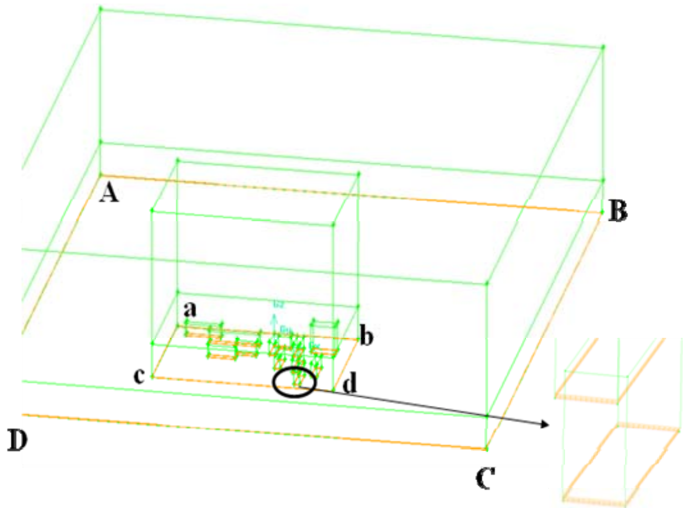
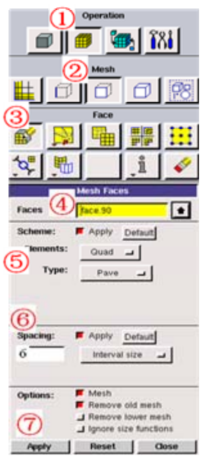


Fig. 32 Define boundary layer mesh for the two bottom faces.

Fig. 33 Boundary layer mesh created at the bottom faces.

2.2.3 Mesh the bottom face of the small domain using Pave

Meshing of the volume can be started with mesh of one of its faces. For this case it is obvious to start with the bottom face, due to the fact that all the buildings locate at the bottom face (the ground). As defined by the size function, the mesh interval close to the buildings is much smaller than the interval in the area away from the buildings. The bottom face has to be meshed with Pave method and quad as mesh element, see Fig. 34. Do the following steps to mesh the bottom face of the volume 'domain small'.



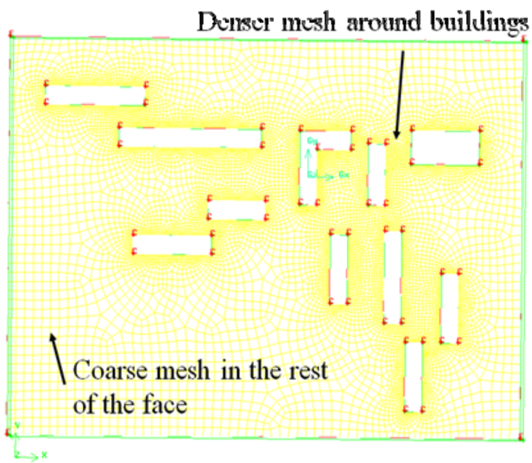


Fig. 34 Mesh the bottom face of the small domain using Pave.

Fig. 35 Boundary layer mesh created at the bottom faces.

① Click pad 'Mesh'

② Click pad 'Face'

③ Mesh faces

④ Select face abdc (in Fig. 33).

⑤ select quad as elements, pave as mesh type.

⑥* Key in an interval size of 6..

⑦ Apply

*⑥, The interval size determines the size of element in meter.

The meshed face is shown in Fig. 35. It can be seen that the mesh around the buildings is denser than the mesh away from the buildings.

2.2.4 Mesh the small domain with Cooper method

The volume 'domain small' can now be meshed with Cooper method.

When meshing using Cooper method, the mesh on the bottom face will be projected along the volume so that a 3D volume mesh is created. The bottom face and the top face are called the source faces.

Do the follows to mesh the small domain:

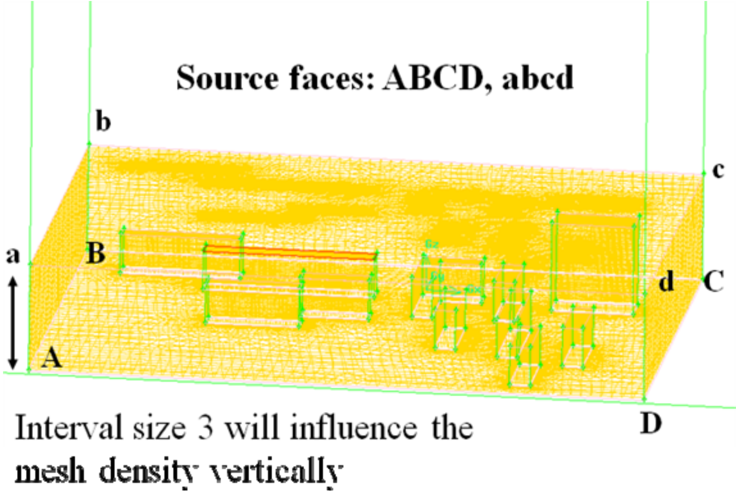


Fig. 36 Mesh the small domain using Cooper method.

Fig. 37 The meshed volume 'domain small'.

① Click pad 'Mesh'

② Click pad 'Volumes'

③ Mesh volumes

④ Select volume 'domain small'.

⑤ select Hex/Wedge as elements, cooper as mesh type.

⑥* Pick sources faces ABCD and abcd (see Fig. 37).

⑦* Key in an interval size of 3.

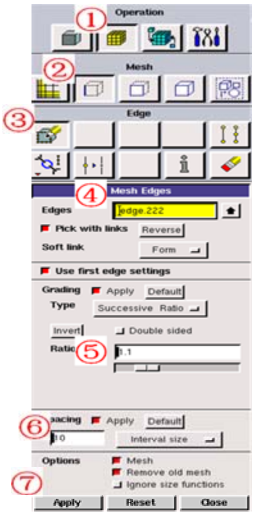
⑧ Apply

Note: *⑥, If applicable Gambit will automatically choose source faces which sometimes can be wrong. It is always a good idea to check the source faces. Manual selection of source faces is needed sometimes.

*⑦, The mesh on the bottom face will be kept if the radio button 'Remove lower mesh' is disabled. The interval size influences the mesh in the vertical direction (building height). Since the idea is to project the mesh on the bottom face ABCD through the volume, the mesh shall not be removed.

2.2.5 Mesh the volume just above 'domain small' with Cooper method

(1) To mesh the volume, the first step is to mesh the four vertical edges of the volume, see Fig. 39.



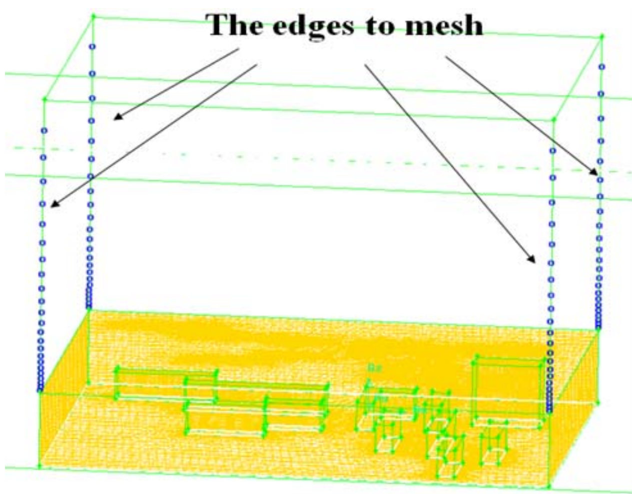


Fig. 38 Mesh edges with uneven intervals.

Fig. 39 The edges to be meshed.

① Click pad 'Mesh'

② Click pad 'Edges'

③ Mesh Edges

④ Select the four edges see Fig. 39.

⑤* Key in a successive ratio of 1.1.

⑥ Key in an average interval size of 10.

⑦ Apply

*⑤, A successive ratio of 1.1 means the mesh interval increase 10%. The direction of the increment can be changed by 'Reverse'.

(2) Mesh the volume.

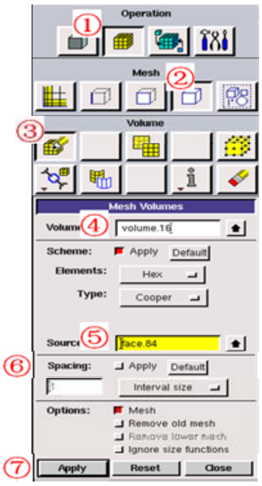


Fig. 40 Mesh the volume just above 'domain small'.

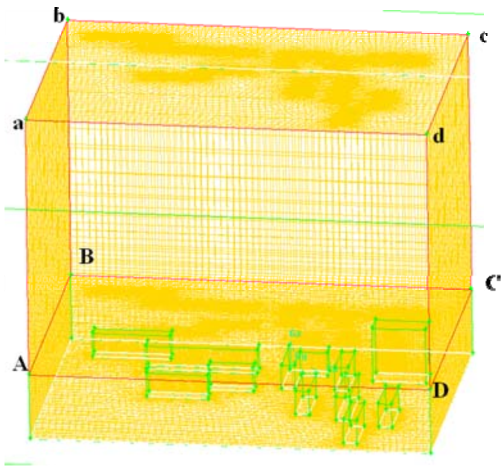


Fig. 41 The mesh volume.

① Click pad 'Mesh'

② Click pad 'Volumes'

③ Mesh volumes

④* Select the volume just above 'domain small'.

⑤* Pick sources faces ABCD and abcd (see Fig. 41).

⑥ Deactivate spacing.

⑦ Apply

Note: *④, Depend on the model you have created, it may not be volume.16 as shown in Fig. 40.

*⑤, It may not be face 84 as shown in Fig. 40.

2.2.6 Mesh the rest of the volumes

The next step is to mesh the bottom outer and top outer volumes. The first step is to mesh the bottom outer face shown as face ABCD in Fig 43. Since the small domain has a denser mesh than the outer domain, it is necessary to assign a size function for face ABCD to control the gradual increase of interval size.

(1) Define size function for the bottom face of the outer volume, see Fig. 42.

① Click pad 'Tools'

② Click pad 'Size function'

③ Create size function

④ Select meshed as type

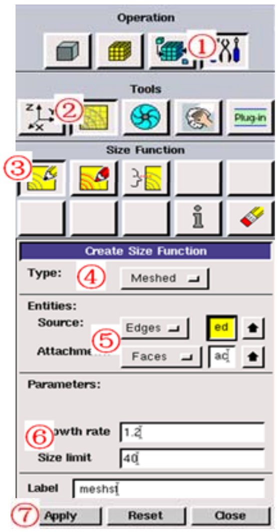
⑤* Select the four inner edges as source edges and the Face ABCD as attachment.

⑥ Key in a growth rate of 1.2 and a size limit of 40.

⑦ Apply

Note: *④, The meshed type size function means that the new mesh will grow from the existing mesh on the sources edges/faces.

*⑤, The new mesh will grow from the mesh on the four inner edges. The new mesh will be attached to the face ABCD.



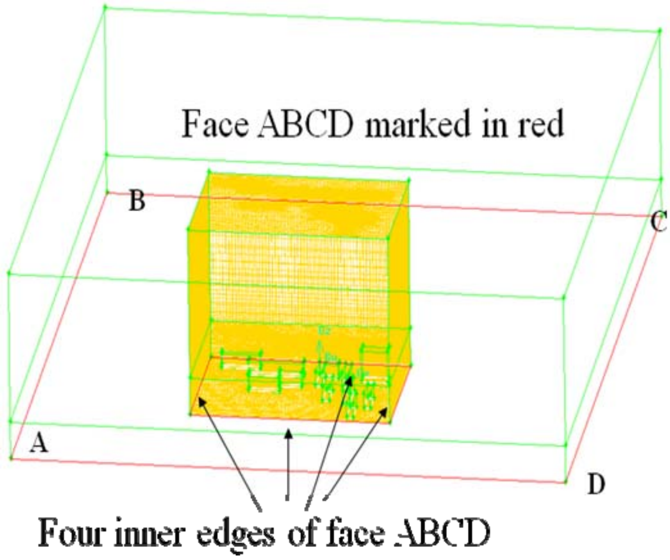
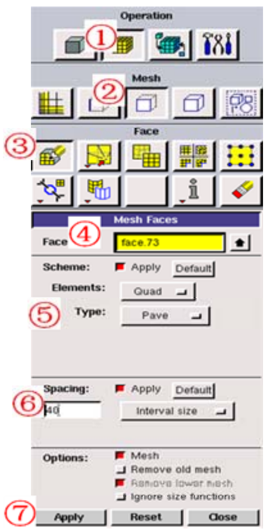


Fig. 42 Define size function for face ABCD in Fig.43.

Fig. 43 Size function with the four inner edges as sources and the Face ABCD as attachment.

(2) Mesh the bottom face of the outer volume.

The face ABCD can now be meshed with Pave method. It can be seen from the meshed face in Fig. 45 that the mesh on the inner four edges is used as basis to grow mesh further in the face ABCD. The interval size grows with a factor of 1.2. In the area away from the four inner edges, the interval size is 40 m.



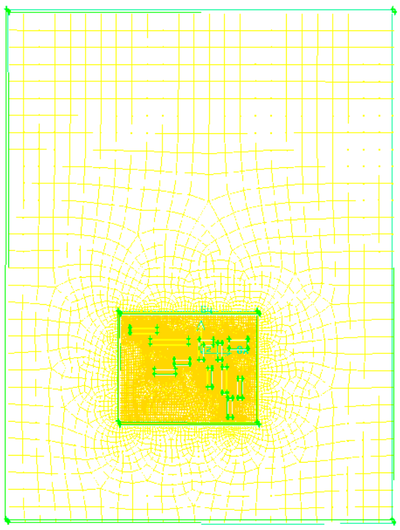


Fig. 44 Mesh the face ABCD (see Fig.43) with Pave method.

Fig. 45 The meshed fac.

① Click pad 'Mesh'
② Click pad 'Faces'
③ Mesh Faces
④ Select the face ABCD as shown in Fig. 43.
⑤ Select quad as elements and Pave as type..
⑥* Key in an interval size of 40.
⑦ Apply

Note: *⑥ The interval size shall be the same as the size limit specified when defining size function.

(3) Mesh the bottom outer volume.

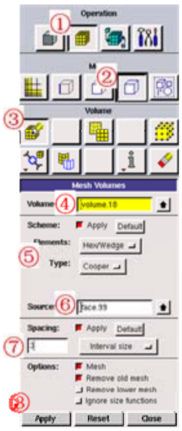


Fig. 46 Mesh the bottom outer volume.

① Click pad ‘Mesh’

② Click pad ‘Volumes’

③ Mesh volumes

④ Select the bottom outer volume

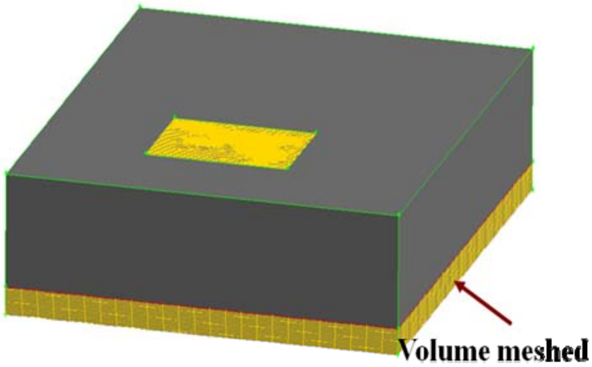


Fig. 47 The meshed volume.

⑤ Select Hex/Wedge as elements and Cooper as type..

⑥* Pick the bottom and the top face as source faces .

⑦* Key in an interval size of 3 m.

⑧ Apply

Note: *⑥ If applicable Gambit will automatically choose source faces which sometimes can be wrong. It is always a good idea to check the source faces. Manual selection of source faces is needed sometimes.

*⑦ The interval size determines the mesh interval in the vertical direction. It shall be the same as the vertical interval size of the inner volume (‘domain small’).

(4) Mesh the upper outer volume.

Since the four edges a, b, c, d are already meshed, the other four edges A, B, C, D needed to be meshed. It is important to have exactly the same mesh on the edges A,B, C, D as the edge a, b, c, d, so that the grid line is horizontal.

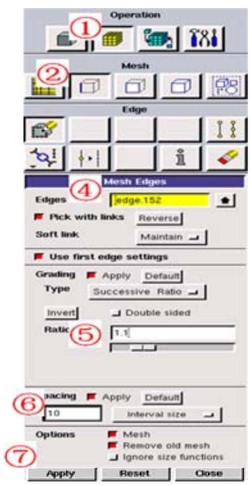


Fig. 48 Mesh the edges A, B, C, D (shown in Fig. 49).

① Click pad ‘Mesh’

② Click pad ‘Edges’

③ Mesh edges

④ Select the four edges A, B, C, D

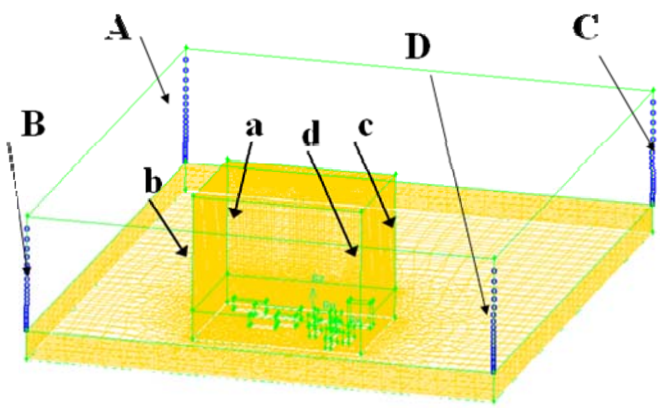


Fig. 49 The meshed edges.

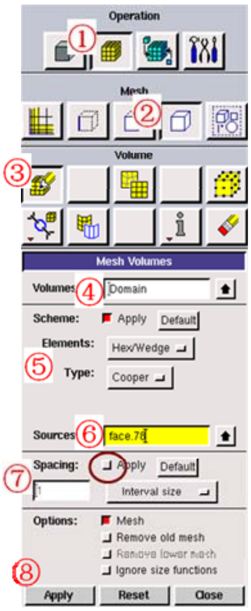
⑤* Key in a ratio of 1.1.

⑥ Key in an average interval of 10.

⑦ Apply

Note: *⑤ The ratio of 1.1 means that the interval size increases from the bottom to the top with a factor of 1.1. The increment direction shall be altered by the radio button ‘reverse’ so that the mesh is denser at the bottom and coarse at the top.

Then mesh the upper outer volume.



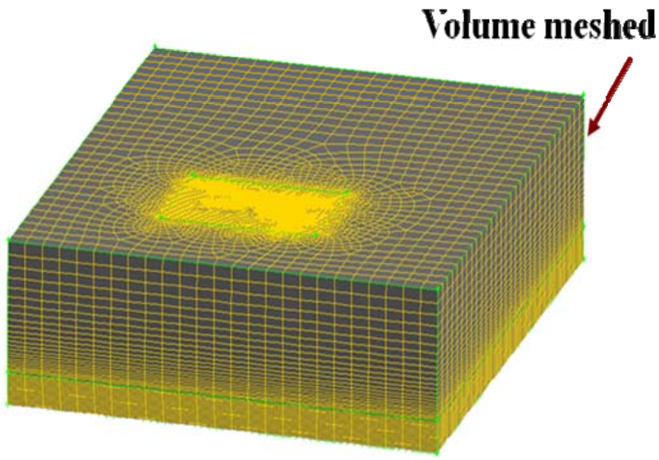


Fig. 50 Mesh the top outer volume.

Fig. 51 The meshed volume.

① Click pad ‘Mesh’

② Click pad ‘Volumes’

③ Mesh volumes

④ Select the top outer volume

⑤ Select Hex/Wedge as elements and Cooper as type..

⑥* Pick source faces.

⑦* Deactivate spacing.

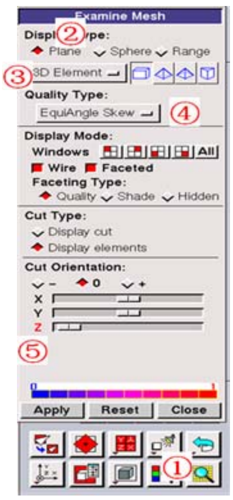
⑧ Apply

Note: *⑥ If applicable Gambit will automatically choose source faces which sometimes can be wrong. It is always a good idea to check the source faces. Manual selection of source faces is needed sometimes.

*⑦ The spacing is deactivated as all the eight side edges are already meshed.

2.2.7 Examine mesh quality

Now the mesh is examined.



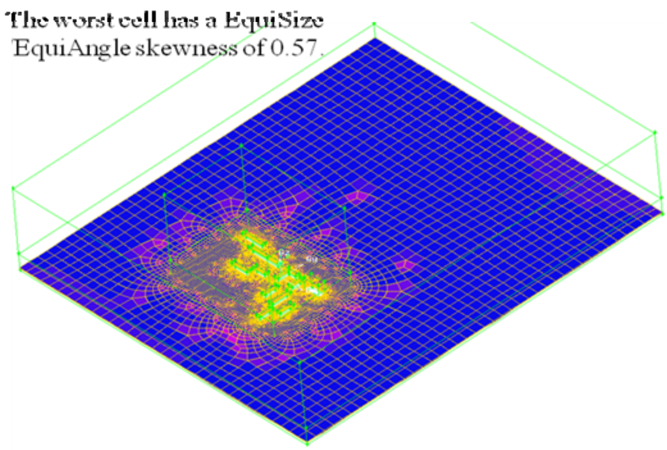


Fig. 52 Examination of mesh quality.

Fig. 53 Mesh quality at a horizontal plane.

① Click pad 'Check mesh'

②* Select Plane or Range

③* Select 3D element, Enable hex.

④ Select EquiAngle skew as quality type.

⑤ Slide the bar of lower and upper limit to check the mesh

Note: *② Plane: to check with a planar view of the mesh; Range: to check the mesh within a range.

*③ Enable the element types to be checked by clicking on the corresponding pad/pads.

3 Specification of boundary conditions

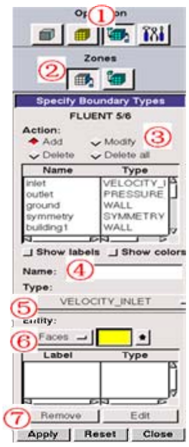


Fig. 54 Boundary condition.

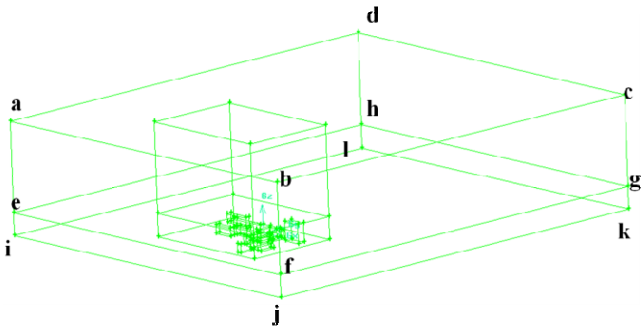


Fig. 55 The model.

① Click Boundary pad

② Specify boundary types

③ Select action Add/Modify/Delete

④ Key in Name.

⑤ Select boundary types

⑥ Select the faces to specify the boundary

⑦ Apply

As shown in Fig. 54, define boundary conditions for the following faces:

Boundary type Velocity inlet for the faces 'abfe' and 'ijfe'.

Boundary type 'Pressure outlet' for the faces 'dcgh' and 'hgkl'.

Boundary type 'Symmetry' for the faces 'abcd', 'adhe', 'ehli', 'bcgf', 'fgkj'.

Boundary type 'Wall' for the bottom faces 'face.73' and 'face.90'.

Boundary type 'Wall' for all the faces of the buildings.

4 Export of the mesh

The geometry and mesh can be exported to a file which later can be read into Fluent.

File → Export → Mesh

Type the name of the mesh file to be exported.

Save as 'Exp2-hex.dbs' and exit Gambit.