Fluid-flow simulation over a computer-generated aircraft surface is generated using inviscid and viscous simulations. A fluid-flow mesh of fluid cells is obtained. At least one inviscid fluid property for the fluid cells is determined using an inviscid fluid simulation that does not simulate fluid viscous effects. A set of intersecting fluid cells that intersects the aircraft surface are identified. One surface mesh polygon of the surface mesh is identified for each intersecting fluid cell. A boundary-layer prediction point for each identified surface mesh polygon is determined. At least one boundary-layer fluid property for each boundary-layer prediction point is determined using the at least one inviscid fluid property of the corresponding intersecting fluid cell and a boundary-layer simulation that simulates fluid viscous effects. At least one updated fluid property for at least one fluid cell is determined using the at least one boundary-layer fluid property and the inviscid fluid simulation.

20 Claims, 12 Drawing Sheets
U.S. PATENT DOCUMENTS

2006/0025973 A1 2/2006 Kim
2008/0175511 A1 7/2008 Kamatauchi
2008/0300835 A1 12/2008 Dixon
2009/0171596 A1 7/2009 Houston
2010/0268517 A1 10/2010 Calmels
2010/0280802 A1 11/2010 Calsmes
2012/0065950 A1 * 3/2012 Lu ................. 703/2

OTHER PUBLICATIONS

Lagree, P. "Interactive Boundary Layer [IBL] or Inviscid-Viscous Interactions [IVI or VII]", Dec. 2009.*


* cited by examiner
Fig. 9

900 Obtain a Fluid-Flow Mesh

902 Determine at Least One Inviscid Fluid Property for Fluid Cell in the Fluid-Flow Mesh

904 Identify Intersecting Fluid Cells of the Fluid-Flow Mesh that Intersect Aircraft Surface

906 Identify One Surface Mesh Polygon for Each Intersecting Fluid Cell

908 Determine at Least One Boundary-Layer Fluid Property

910 Determine at Least One Updated Fluid Property
Fig. 10

1052 Determine Boundary-Layer Prediction Point Corresponding to an Intersecting Fluid Cell

1054 Determine at Least One Boundary-Layer Fluid Property for the Boundary-Layer Prediction Point

1055 Pass the at Least One Boundary-Layer Fluid Property to Inviscid CFD Simulation Module

1050 Boundary-Layer CFD Simulation Module

1012 Determine at Least One Inviscid Fluid Property of an Intersecting Fluid Cell

1014 Pass the at Least One Inviscid Fluid Property to a Corresponding Boundary-Layer Prediction Point

1016 Determine at Least One Updated Fluid Property of the Intersecting Fluid Cell

1010 Inviscid CFD Simulation Module
GENERATING INVISCID AND VISCOUS FLUID-FLOW SIMULATIONS OVER AN AIRCRAFT SURFACE USING A FLUID-FLOW MESH

STATEMENT REGARDING FEDERALLY SPONSORED RESEARCH OR DEVELOPMENT

This invention was made with Government support under contract NNL08AA08C awarded by NASA. The Government has certain rights in the invention.

BACKGROUND

1. Field

This application relates generally to simulating a fluid flow over an aircraft surface and, more specifically, to generating both inviscid and viscous fluid-flow simulations using a fluid-flow mesh.

2. Description of the Related Art

Aerodynamic analysis of an aircraft moving through a fluid typically requires an accurate prediction of the properties of the fluid surrounding the aircraft. Accurate aerodynamic analysis is particularly important when designing aircraft surfaces, such as the surfaces of a wing or control surface. Typically, the outer surface of a portion of the aircraft, such as the surface of a wing, is modeled, either physically or by computer model, so that a simulation of the fluid flow can be performed and properties of the simulated fluid flow can be measured. Fluid-flow properties are used to predict the characteristics of the wing including lift, drag, boundary-layer velocity profiles, and pressure distribution. The flow properties may also be used to map laminar and turbulent flow regions near the surface of the wing and to predict the formation of shock waves in transonic and supersonic flow.

A computer-generated simulation can be performed on a computer-generated aircraft surface to simulate the fluid dynamics of a surrounding fluid. The geometry of the computer-generated aircraft surface is relatively easy to change and allows for optimization through design iteration or analysis of multiple design alternatives. A computer-generated simulation can also be used to study situations that may be difficult to reproduce using a physical model, such as sonic flight conditions. A computer-generated simulation also allows a designer to measure or predict fluid-flow properties at virtually any point in the model by direct query, without the difficulties associated with physical instrumentation or data acquisition techniques.

In some cases, a computer-generated simulation includes a computational fluid dynamics (CFD) simulation module used to predict the properties of the fluid flow. A CFD simulation module estimates the properties of a simulated fluid flow by applying an algorithm that estimates the interaction between small simulated fluid volumes, also referred to as fluid cells. Because a single CFD simulation module may include millions of individual fluid cells, the complexity of the relationship between fluid cells can have a large effect on the computational efficiency of the simulation. Complex CFD simulation modules can be computationally expensive and require hours or even days to execute using high-performance computer processing hardware.

To reduce the computational burden, in some instances it is desirable to use a CFD simulation module that simplifies the fluid dynamics and produces a fluid simulation that can be solved more rapidly. For example, for fluid flows that are relatively uniform or are located away from an aircraft surface, a simplified simulation that minimizes or ignores fluid properties that have little effect on the overall behavior of the fluid can be used. In this way, processing time is improved without sacrificing accuracy or resolution of the final results.

In other situations, where the fluid flow is not as uniform, it may be necessary to use a CFD simulation module that is more sophisticated and capable of accurately predicting the fluid properties, using more complex fluid dynamics. However, more sophisticated simulation modules are also likely to require more computing resources and therefore require more time to solve.

It may be advantageous to construct a hybrid computer-generated simulation that employs both a simplified CFD simulation module in locations where the fluid flow is relatively uniform, and a more sophisticated CFD simulation module in locations where the fluid dynamics are more complex. By combining different CFD simulation modules, a hybrid computer-generated simulation may increase processing speed while producing accurate results.

Using multiple CFD simulation modules may be difficult, particularly if the CFD simulation modules were not initially designed to work together. The interface between the simulation modules must be constructed so that the resulting computer-generated simulation is both computationally efficient and analytically robust. The techniques described herein solve some of the difficulties in implementing a computer-generated simulation using multiple simulation modules.

SUMMARY

One exemplary embodiment includes a computer-implemented method of generating a fluid-flow simulation over a computer-generated aircraft surface, the computer-generated aircraft surface comprised of a surface mesh of surface mesh polygons. A fluid-flow mesh is obtained for simulating a fluid flow over the aircraft surface, the fluid-flow mesh comprising a plurality of fluid cells. At least one inviscid fluid property for each of the fluid cells is determined using an inviscid fluid simulation that does not simulate fluid viscous effects. A set of intersecting fluid cells, of the plurality of fluid cells, that intersects the aircraft surface is identified. One surface mesh polygon of the surface mesh for each intersecting fluid cell of the set of intersecting fluid cells is also identified. A boundary-layer prediction point for each identified surface mesh polygon is determined. At least one boundary-layer fluid property for each boundary-layer prediction point is determined using the at least one inviscid fluid property of the corresponding intersecting fluid cell and a boundary-layer simulation that simulates fluid viscous effects. At least one updated fluid property is determined for at least one fluid cell of the plurality of fluid cells using the at least one boundary-layer fluid property and the inviscid fluid simulation.

DESCRIPTION OF THE FIGURES

FIG. 1 depicts a computer-generated fluid flow applied to a computer-generated aircraft surface.

FIGS. 2a and 2b depict an exemplary fluid flow around a wing surface.

FIG. 3 depicts an exemplary quadrilateral surface mesh and a corresponding structured mesh of the fluid flow.

FIG. 4 depicts an exemplary surface mesh and fluid-flow mesh.

FIG. 5 depicts an exemplary fluid-flow mesh and boundary-layer prediction points.

FIG. 6 depicts a cross-sectional view of an exemplary fluid-flow mesh and boundary-layer prediction points.
As discussed above, a computer-generated simulation can be used to analyze the aerodynamic performance of a proposed aircraft surface, such as a wing or control surface. Using known geometry modeling techniques, a computer-generated aircraft surface that represents the outside surface of the proposed aircraft can be constructed. FIG. 1 depicts an exemplary computer-generated aircraft surface of the Space Shuttle orbiter vehicle, external tank, and twin solid rocket boosters. A CFD fluid simulation module has been applied using the computer-generated aircraft surface of the Space Shuttle orbiter to predict the fluid properties of an exemplary fluid flow.

As shown in FIG. 1, the results of the simulation can be visually represented as shaded regions on the computer-generated aircraft surface of the Space Shuttle. Different shades represent the predicted pressure distribution resulting from the simulated fluid flow. In FIG. 1, transitions between the shaded regions represent locations of predicted pressure change across the surface of the Space Shuttle. Similarly, different pressures in the surrounding fluid flow are represented as differently shaded regions.

In FIG. 1, the simulation of the fluid flow is visualized by depicting the predicted pressure distribution. However, the simulation may be visualized using other fluid properties, including surface velocity, air temperature, air density, and others. Additionally, the simulation may be used to visualize locations of developing shock waves or transitions between laminar and turbulent flow.

The simulation allows the designer or engineer to evaluate the performance of the aircraft geometry for various flow conditions. If necessary, changes can be made to the aircraft geometry to optimize performance or eliminate an unwanted aerodynamic characteristic. Another simulation can be performed using the modified geometry, and the results can be compared. To allow for multiple design iterations, it is advantageous to perform multiple simulations in a short amount of time. However, as described above, there is a tradeoff between speed and accuracy of the simulation depending on the type of CFD simulation module used.

Typically, a computer-generated simulation represents a fluid flow as a three-dimensional fluid-flow mesh of small volumes of fluid called fluid cells. As discussed in more detail below, the shape of the fluid cells can vary depending on the method used to construct the fluid-flow mesh. A CFD simulation module predicts the interactions between the fluid cells in the fluid-flow mesh, using a fundamental algorithm or field equation.

The speed and accuracy of a CFD simulation module depends, in part, on the field equation used to predict the interaction between the flow cells. In some instances, the field equation simplifies the relationship between flow cells by ignoring or minimizing certain dynamic contributions. These field equations are typically less complex, and therefore are more computationally efficient. For instance, a simplified algorithm called the Euler method may be used to simulate a fluid flow when viscous effects can be minimized or ignored.

Viscous effects of a fluid can be ignored when, for example, there is not a significant velocity difference between adjacent fluid cells, and therefore shear forces due to internal friction or viscosity are minimal. A CFD simulation module that ignores or minimizes effects of fluid viscosity can also be referred to as an inviscid simulation.

In other instances, a more complex field equation is used to more accurately predict the interaction between the fluid cells. For example, a Navier-Stokes method can be used to simulate the pressure and shear forces on the flow cells. Unlike the Euler method mentioned above, the Navier-Stokes equation accounts for the effects of viscosity and offers a more accurate simulation of a fluid flow. A simulated fluid flow that accounts for effects due to fluid viscosity can also be referred to as a viscous simulation.

However, the improved accuracy of the Navier-Stokes method comes at the cost of increased computational load, and therefore the Navier-Stokes method is generally slower to compute than an Euler-based algorithm. Thus, selecting the field equation for a CFD module often involves a tradeoff between speed and accuracy. In practice, designers may use faster Euler-based CFD models to evaluate multiple design iterations and then validate the final design iteration with a more accurate Navier-Stokes-based CFD model. However, if the Navier-Stokes CFD simulation reveals a design problem, the entire process must be repeated, wasting valuable time and computing resources.

The techniques described below are computer-generated simulations that use multiple algorithms to achieve acceptable accuracy without requiring the computational burden of a full Navier-Stokes CFD simulation. In many simulations, there is a portion of the flow that can be accurately predicted without taking viscous contributions into account. For example, portions of the fluid flow that are located away from an aircraft surface, such as a wing surface, have a relatively uniform velocity profile. Therefore, an inviscid simulation using, for example, an Euler-based analysis, can be used to accurately predict the behavior of these regions of the fluid flow. In other locations of the fluid flow where there is a less uniform velocity profile, a more complex, viscous simulation can be used.

The techniques described herein provide a method of generating a simulation using more than one field equation to simulate the fluid flow over a computer-generated aircraft surface using a fluid-flow mesh and surface mesh. If the values produced by multiple simulations are not passed between the meshes by using a one-to-one correlation between mesh elements, error and instability may be introduced into the computer model. Many of these errors may be overcome or greatly reduced by establishing a one-to-one correlation between, for example, an inviscid fluid-flow mesh element and surface (or boundary-layer) mesh element. The following technique provides one example of how a one-to-one correlation can be maintained for computer models using a fluid-flow mesh and a surface mesh that do not align.
The following discussion provides an example of a simulated fluid flow over an aircraft surface, such as a wing surface. The technique may also be applied to a simulated fluid flow over any type of surface subjected to a fluid flow.

1. Simulating Fluid Flow Over A Wing

FIGS. 2a and 2b depict a two-dimensional representation of a fluid flow over a wing surface 202 classified by two regions: a free stream region 204 and a boundary-layer region 206. As shown in FIGS. 2a and 2b, the boundary-layer region 206 is located near a wing surface 202 and is characterized by a sharply increasing velocity profile 208. Skin friction causes the fluid very close to the wing surface 202 to be essentially zero, with respect to the surface. A sharply increasing velocity profile develops as the velocity increases from a near-zero velocity to the free stream velocity. The sharply increasing velocity profile 208 in the boundary-layer region 206 creates shear forces within the boundary-layer fluid flow. Due to the internal shear forces, viscous properties of the fluid influence the boundary-layer fluid flow. Therefore, a simulation of the flow in the boundary-layer region 206 should account for viscous contributions to the flow dynamics. In some cases, the fluid in boundary-layer region 206 may be characterized as turbulent flow (region 210). Due to fluid voracity, viscous properties of the fluid influence the fluid flow. Thus, a simulation of the turbulent flow should also account for viscous contributions to the flow dynamics. For purposes of this discussion, laminar and turbulent regions are treated as one boundary-layer region and simulated using a single CFD simulation module.

A CFD simulation module that accounts for viscosity may also be called a viscous CFD simulation module or a boundary-layer CFD simulation module. Below, exemplary field equations for a boundary-layer CFD simulation module are provided according to a Drela boundary-layer technique. Drela, M. “XFOIL: An Analysis and Design System for Low Reynolds Number Airfoils,” pp. 1-12, Proceedings of the Conference on Low Reynolds Number Aerodynamics (T. J. Mueller ed., Univ. of Notre Dame, Notre Dame, Ind., 1989).

Equation 1, below, represents a boundary-layer integral momentum equation for compressible flow:

$$\frac{d\theta}{dx} + (2 + H - M_e^2) \frac{\theta}{u_e} \frac{du_e}{dx} = \frac{C_f}{2}.$$  
Equation 1

where $\theta$ is the momentum thickness, $H$ is the shape factor, $M_e$ is the boundary-layer edge Mach number, $u_e$ is the boundary-layer edge velocity, and $C_f$ is the skin friction coefficient.

Equation 2, below, represents a boundary-layer kinetic energy integral equation:

$$\frac{dH}{dx} + (2H^* + H^*(1 - H)) \frac{\theta}{u_e} \frac{du_e}{dx} = 2C_p - H^* \frac{C_f}{2}.$$  
Equation 2

As used in equations 1 and 2, above, shape factors $H$, $H^*$, and $H^{**}$ are defined as:

$$H = \frac{\theta}{\theta_H} = \frac{\theta}{\theta^*} = \frac{\theta^{**}}{\theta^*};$$

Solving equations 1 and 2 for local velocity $u$ and density $\rho$, the boundary-layer CFD simulation module can predict the fluid properties for portions of the fluid flow within the boundary-layer region 206. Additionally, characteristics of the boundary layer, including boundary-layer thickness, can also be determined once the fluid properties are known.

The portions of the fluid flow outside of the boundary-layer region 206 may be designated as a free stream region 204. The free stream region 204 is typically located away from the wing surface 202. However, the free stream may be close to the wing surface 202 in areas where the boundary layer is thin or has yet to develop. See, for example, the portion of the fluid flow in FIG. 2a near the leading edge of the wing surface 202. The free stream region 204 is usually characterized as having a relatively uniform velocity profile 212. When there is a uniform velocity profile 212, internal shear forces acting on a fluid may be relatively small, and therefore viscous contributions to the fluid dynamics can be minimized or ignored.

A CFD simulation module that ignores viscous effects may also be called an inviscid CFD simulation module. Equation 3, below, provides an exemplary field equation for an inviscid CFD simulation module. Equation 3, also called the Euler method, represents the conservation of mass, conservation of three components of momentum, and conservation of energy:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0.$$  
Equation 3

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \frac{1}{\rho} \frac{\partial \tau_x}{\partial x} + \frac{1}{\rho} \frac{\partial \tau_y}{\partial y} + \frac{1}{\rho} \frac{\partial \tau_z}{\partial z}.$$  
Equation 3
where u, v, and w are components of the velocity vector, p is the pressure, ρ is the density, and E is the total energy per unit volume. Combining Equation 3 with an equation of state (e.g., the ideal gas law), an inviscid CFD simulation module can predict the fluid properties for the free stream fluid region.

2. Combining Multiple CFD Simulation Modules

Using the techniques described below, both the boundary-layer and free stream regions can be simulated by combining viscous and inviscid CFD simulation modules. For example, FIG. 3 depicts a fluid flow represented as a structured mesh of inviscid fluid cells 302 and a structured mesh of boundary-layer fluid cells 304. The mesh of inviscid and boundary-layer fluid cells 302 and 304 are depicted in FIG. 3 in a cross-sectional representation (two dimensional). Note, however, that the fluid cells are actually three-dimensional volumes of fluid.

FIG. 3 also depicts a surface mesh 306 of quadrilateral polygons representing the surface of a wing. The surface mesh 306 should approximate the curvilinear shape of the wing surface without creating gaps or breaks between quadrilateral polygons. For relatively simple wing surfaces as shown in FIG. 3, the mesh can be created from multiple wing surface cross-sectional profiles, where each wing profile is approximated by short line segments. The quadrilateral polygons are created by connecting the vertex of each short line segment for adjacent wing profiles.

The structured mesh of inviscid fluid cells 302 shown in FIG. 3 is a mesh of fluid volumes defined using a set of vertices of the surface mesh 306. For a set of four adjacent vertices on the surface mesh 306, a volume 310 is projected from the surface of the wing in a direction as close to a surface normal as possible. The volume is partitioned into fluid cells 308 by defining at least two surfaces 312 that offset a given distance from the surface of the wing. The structured mesh depicted in FIG. 3 does not intersect the surface of the wing represented by the surface mesh 306.

In FIG. 3, the surface mesh 306, the structured mesh of boundary-layer fluid cells 304, and the structured mesh of inviscid fluid cells 302 have been created so that there is a one-to-one correlation to cells at the mesh boundaries. That is, each mesh element that borders another mesh corresponds to exactly one cell of the bordering mesh. Therefore, for a given polygon in the surface mesh, there is one corresponding boundary-layer fluid cell, and for that boundary-layer fluid cell there is one corresponding inviscid fluid cell. This arrangement is advantageous in that it allows data to be passed from one cell to another without having to interpolate or estimate the closest neighboring cells.

The construction of the meshes shown in FIG. 3 typically requires that the surface mesh 306 of the wing be comprised entirely of quadrilateral polygons. The structured mesh of boundary-layer fluid cells 304 and the structured mesh of inviscid fluid cells 302 are then constructed using the vertices of the surface mesh 306 as starting points.

There are, however, drawbacks to using this meshing technique. First, it can be difficult to apply a quadrilateral surface mesh to complex surface geometries. For example, the segmented-line profile technique described above does not work for surfaces without a relatively uniform longitudinal cross section. Also, complex geometries created by intersections between surfaces can be difficult to model using a quadrilateral mesh. For example, a pylon and nacelle hanging off the leading edge of a wing may be difficult to automate and typically requires human interaction or troubleshooting to create a continuous, gap-free surface mesh.

FIG. 4 also depicts another technique for creating surface and fluid-flow meshes. In FIG. 4, the inviscid portion of the fluid flow is represented by a fluid-flow mesh 402, which is depicted as being a Cartesian mesh. A Cartesian mesh is defined as a mesh of cube or rectangular cuboid fluid cells 404. That is, each fluid cell 404 is bounded by six flat faces where opposite faces are parallel and adjacent faces are orthogonal. In some cases, larger fluid cells are divided into smaller fluid cells by defining additional faces at the midpoint of the existing faces. Thus, the fluid cells in a Cartesian mesh can be different sizes. In the present exemplary embodiment, because the fluid-flow mesh 404 is composed entirely of cube volumes, fluid-flow mesh 404 can be automatically generated minimizing human interaction and troubleshooting. Additionally, cube cells tend to produce less error when using CFD simulation techniques due, in part, to the uniformity of the mesh cells in multiple mesh directions.

FIG. 4 also depicts a surface mesh 406 that is constructed using triangular mesh elements rather than quadrilateral mesh elements. Triangular mesh elements are better suited for representing complex geometries or intersecting surfaces. This technique is also called triangulating the wing surface and can be automated for complex geometries with little or no human intervention.

Compared to the meshing shown in FIG. 3, the meshing technique depicted in FIG. 4 may be easier to create using automated methods and may result in a more consistent, continuous mesh. However, using the meshing technique depicted in FIG. 4, there is no longer a one-to-one correlation between neighboring mesh elements. For example, a given fluid cell in the fluid-flow mesh of boundary-layer fluid cells 404 no longer corresponds to a fluid cell in the fluid-flow mesh 402. A given fluid cell in the fluid-flow mesh of boundary-layer fluid cells 404 also does not correlate to a single surface mesh element on the surface mesh 406 of the wing surface. Additionally, for portions of the fluid-flow mesh 402 near the wing surface, there will be at least some fluid cells that partially intersect one or more of the triangulated surface mesh elements in the surface mesh 406 of the wing surface. Therefore, for any one boundary cell, there are multiple candidate neighboring cells that can be used to exchange data.

Without a one-to-one correlation between cells, passing data between the different mesh elements is more complicated and more prone to error. For example, values from one or more bordering inviscid fluid cells may be passed to a neighboring boundary-layer fluid cell. One or more inviscid fluid cells must be selected and the values interpolated depending on the degree of overlap between the cells. These techniques tend to introduce error into the simulation and reduce the robustness of the solution.
In some cases, the boundary-layer mesh can be derived from the intersection of the fluid-flow mesh and the surface mesh. As shown in FIGS. 5 and 6, a set of boundary-layer fluid cells may be defined according to the location of surface mesh elements with respect to an intersecting plane of fluid cells of the fluid-flow mesh. FIG. 5 depicts a surface mesh of the wing surface intersected by a two-dimensional strip representing the centerline of a plane of fluid cells of the fluid-flow mesh. A boundary-layer prediction point is created at the intersection of the strip and the border of each intersected surface mesh element. The boundary-layer prediction points can be used to construct a local boundary-layer fluid-flow mesh with a boundary-layer fluid cell centered on each boundary-layer prediction point. The resulting boundary-layer fluid cells can be used to simulate the boundary-layer portion of the fluid flow. Not every boundary-layer fluid cell is given a plane cell as viewed along the wing cross-section. See, for example, FIG. 6 depicting boundary-layer fluid cells centered on boundary-layer prediction points.

Using the boundary-layer fluid cells, a boundary-layer CFD simulation module can be coupled with an inviscid CFD simulation module to predict the fluid property values on and around the wing. In one example, an inviscid CFD simulation module determines a local pressure, fluid density, and local velocity values for each fluid cell in the fluid-flow mesh. The boundary-layer CFD simulation module receives fluid property values from neighboring inviscid fluid cells. If there is more than one neighboring inviscid fluid cell present, the boundary-layer CFD module typically interpolates the inviscid fluid properties to combine the results of the multiple neighboring inviscid fluid cells. As discussed above, interpolation techniques may introduce error into the simulation due to errors in the interpolation and approximating the contributions from multiple inviscid fluid cells.

Using, for example, equations 1 and 2 above, the boundary-layer CFD module determines the properties of the simulated fluid flow in the boundary-layer region. The boundary-layer CFD simulation module returns fluid properties and/or a transpiration flux value representing a fictitious fluid flow into or out of the wing. As mentioned above, there may be more than one neighboring inviscid fluid cell that receives these values from a corresponding boundary-layer fluid cell. It is also likely that there is more than one boundary-layer fluid cell neighboring a bordering inviscid fluid cell. This mismatch between flow cells may generate additional error and/or model instability.

Additional problems may arise because the boundary-layer prediction points are not evenly spaced along the two-dimensional strip representing the centerline of a plane of fluid cells of the fluid-flow mesh. As shown in FIGS. 5 and 6, boundary-layer fluid cells, which are centered on the boundary-layer prediction points, are also unequally spaced, resulting in gaps or bunching in the boundary-layer fluid-flow mesh. In fact, the spacing of the boundary-layer fluid cells is often much finer than the spacing of the inviscid fluid-flow mesh. In practice, the mismatched spacing requires numerical smoothing or averaging to obtain more consistent boundary-layer CFD simulation module results. However, the smoothing can introduce further error in the simulation and in some cases may produce false or inaccurate predictions of the fluid behavior.

In summary, because the values produced by each CFD simulation module are not passed to a corresponding fluid cell with a one-to-one correlation, error and instability are introduced into the computer model. Error due to interpolation and smoothing also tends to become exacerbated as the inviscid and boundary-layer CFD simulation modules are iterated multiple times.

As suggested above, many of these errors may be overcome or greatly reduced by establishing a one-to-one correlation between an inviscid fluid cell and a boundary-layer fluid cell. The following technique provides one example of how a one-to-one correlation can be maintained for computer models using an inviscid fluid-flow mesh and a surface mesh that do not align.

3. Multiple CFD Simulation Modules with One-to-One Fluid Cell Correlation

FIG. 9 depicts an exemplary process for simulating a fluid flow using both an inviscid CFD simulation module and a viscous CFD simulation module. Using the process, a one-to-one correlation is determined between the fluid cells used by each CFD module.

In step 702, a fluid-flow mesh is obtained for simulating the inviscid portion of the fluid flow. In one exemplary embodiment, as shown in FIG. 4, the fluid-flow mesh is a Cartesian mesh generated around an aircraft surface, such as a wing surface. The Cartesian mesh has cubic or cuboid fluid cells. Typically, the Cartesian mesh surrounds the aircraft surface, but it is not necessary to do so.

In step 704, an inviscid CFD simulation module is used to determine inviscid fluid properties for each fluid cell in the fluid-flow mesh. The inviscid CFD simulation module may use the Euler-based method in equation 3 to determine the inviscid fluid properties. The inviscid fluid properties include but are not limited to: local velocity vector, fluid pressure, and fluid density.

In step 706, fluid cells of the fluid-flow mesh that intersect the surface of the aircraft surface are identified. For example, FIG. 7 depicts a cross section of a wing surface (an exemplary aircraft surface). The cross section depicted in FIG. 7 is taken along the center of a plane of fluid cells 408 of fluid-flow mesh 402, which is depicted as being a Cartesian mesh. As shown in FIG. 7, some of the fluid cells 408 of the fluid-flow mesh 402 intersect the surface of the wing. Fluid cells that at least partially intersect the wing surface are designated as intersecting fluid cells 710.

While FIG. 7 depicts a cross section of a cut created by the intersection of the plane of fluid cells 408 and the wing surface, a similar cut can be created using nearly any arbitrary plane that intersects the wing surface. In some cases, the cut may be defined as the intersection between a spherical or cylindrical surface and the wing surface.

In step 708, one representative surface mesh polygon is identified for each intersecting fluid cell. The representative surface mesh polygon should be selected based on its proximity to the intersecting fluid cell. FIG. 8 depicts the same set of intersecting fluid cells 710 intersecting a surface mesh 406 representing the wing surface shown in FIG. 7. A two-dimensional strip 802 represents the intersection of the center of a plane of fluid cells of the fluid-flow mesh and the surface mesh 406.

In some cases, a centroid of each partially intersecting fluid cell 710 is defined. The centroid 708 is the geometric centroid of the portion of the intersecting fluid cell 710 that is outside the wing surface. A centroid 808 is also defined for nearby surface mesh polygons. A surface mesh polygon having a centroid 808 that is closest to the intersecting fluid cell centroid 708 is selected as the representative surface mesh polygon 804. In this example, the centroid 808 of the representative surface mesh polygon 804 is associated with a point on the plane of fluid cells of the fluid-flow mesh intersecting...
the surface mesh 406. The associated point for each respective selected surface mesh polygon is used as the boundary-layer prediction point 820.

To determine the associated point to be used as the boundary-layer prediction point 820, the intersection of the plane of fluid cells of the fluid-flow mesh and the surface mesh may be represented by an intersection line composed of a series of short line segments. Each line segment represents the intersection between the plane of fluid cells and an intersected surface mesh polygon. The end of each segment falls either on the edge of an intersecting fluid cell or an intersected surface mesh polygon. For a given centroid 808 of a representative surface mesh polygon 804, a line segment is identified that has a midpoint that is closest to the given centroid 808. This midpoint is then used as the associated boundary-layer prediction point 820.

A boundary-layer fluid cell 810 may be centered on each boundary-layer prediction point 820. Using this one-to-one correlation between intersecting inviscid fluid cells and boundary-layer fluid cells, simplifying the data transfer between the two simulations.

In step 910, at least one boundary-layer fluid property is determined for each boundary-layer prediction point 820 or boundary-layer fluid cell 810. The at least one boundary-layer fluid property for a boundary-layer prediction point 820 or boundary-layer fluid cell 810 is determined using a viscous or boundary-layer CFD simulation module and the inviscid fluid properties of the corresponding intersecting fluid cell 710. For example, field equations 1 and 2 described above can be used to determine a momentum thickness using inviscid fluid properties of the corresponding intersecting fluid cell 710.

In some cases, the at least one boundary-layer fluid property includes a boundary-layer thickness value and corresponding transpiration flux value. The boundary-layer thickness value represents the distance from the surface of the aircraft where the fluid flow can be treated as inviscid. For example, as discussed above, the fluid flow may be treated as inviscid if the velocity profile of the fluid flow is uniform enough to ignore the viscosity of the fluid.

A transpiration flux value can also be used to approximate the thickness of the boundary layer by introducing a fictitious flow of air out of the aircraft surface over an arc length along the boundary-layer strip solution. The introduction of the fictitious flow modifies the inviscid simulated flow near the aircraft surface so as to approximate the presence of a boundary-layer flow having an appropriate thickness. As the magnitude of the transpiration flux increases, the fictitious flow increases, simulating a thicker boundary layer. In some cases, the transpiration flux can be used to create a fictitious flow of air into the aircraft surface (negative flux), thereby reducing the thickness of the boundary layer.

The transpiration flux can be determined using the output of the Drela boundary-layer technique described in equations 1 and 2, above. For example, the transpiration flow velocity of the transpiration flux can be determined using:

\[ W_{\text{tr}} = \frac{1}{\rho_{\text{in}}} \frac{d}{ds}(\rho_{\text{in}}U_{\text{in}}e^{s^8}), \]

where \( \rho_{\text{in}} \) is the density of the fluid flow at the aircraft surface, \( U_{\text{in}} \) is the velocity of the fluid flow at the aircraft surface, \( s^8 \) is the computed boundary layer displacement thickness, and \( s \) is the arc length along the boundary-layer strip solution.

Equation 4 is taken from Lock, R. C., and Williams, B. R., “Viscous-Inviscid Interactions in External Aerodynamics,” Proc. Aerospace Sci., Vol. 24, 1987, pp. 51-171. Thus, the transpiration mass flux (density \( \rho_{\text{in}} \) times the transpiration flow velocity \( W_{\text{tr}} \)) is equal to the rate of change of the product of the local density \( \rho_{\text{in}} \), local velocity \( U_{\text{in}} \), and boundary layer displacement thickness \( s^8 \) along the solution strip. A finite difference method can be used to compute the derivative in equation 4. For example, the neighboring solution points along the boundary-layer strip can be used with a second order, backward Lagrange polynomial formulation to compute the derivatives.

In step 912, at least one fluid property of at least one fluid cell of the fluid-flow mesh is updated to account for the changes in the boundary-layer fluid flow. For example, as described above, the transpiration flux introduces a fictitious fluid flow out of the aircraft surface. An inviscid CFD simulation module can then be used to update the fluid properties of the fluid cells of the fluid-flow mesh based on the fictitious fluid flow introduced by the transpiration flux.

As described above in reference to FIGS. 7 and 8, a set of intersecting fluid cells are identified that intersect a surface mesh 406 representing the aircraft surface. In operation 1012, the inviscid CFD simulation module 1010 determines at least one inviscid fluid property of at least one intersecting fluid cell 710 (FIGS. 7 and 8) of the set of intersecting fluid cells.

In operation 1052, one boundary-layer prediction point 820 (FIG. 8) is determined based on the location of the intersecting fluid cell 710 (FIG. 8). As described above, for each selected intersecting fluid cell 710 (FIG. 8), one representative triangulated surface mesh polygon 804 (FIG. 8) is selected based on the proximity of the surface mesh polygon 804 (FIG. 8). A point in the representative triangulated surface mesh polygon, such as the centroid 808 (FIG. 8), is then associated with a point on the plane created by the intersecting fluid cells. The associated point can be designated as a boundary-layer prediction point 820 (FIG. 8), and can be used to define a boundary-layer fluid cell 810 (FIG. 8). Thus, for each intersecting fluid cell, there is one corresponding boundary-layer fluid cell.

In operation 1014, at least one inviscid fluid property of the intersecting fluid cell 710 (FIG. 8) is passed to the corresponding boundary-layer prediction point 820 (FIG. 8) or boundary-layer fluid cell 810 (FIG. 8). The at least one inviscid fluid property includes, for example, local pressure, fluid density, and local velocity values.

In operation 1054, the boundary-layer CFD simulation module 1050 uses the at least one inviscid fluid property to predict the at least one boundary-layer fluid property for the boundary-layer prediction point 820 (FIG. 8) or boundary-layer fluid cell 810 (FIG. 8). As described above, the boundary-layer CFD module 1050 may use the field equations 1 and 2 above to predict one or more boundary-layer fluid proper-
requirements, the results of the boundary-layer CFD simulation module can then be used to determine a boundary-layer prediction point for each boundary-layer fluid cell. In operation 1056, one or more boundary-layer fluid properties are passed to the inviscid simulation module 1010. As described above, in some cases a transpiration flux value can be used to introduce a fictitious flow back into the inviscid fluid flow cell.

In operation 1016, the inviscid CFD simulation module 1010 uses the one or more boundary-layer fluid properties to determine at least one updated fluid property of at least one fluid cell of the fluid-flow mesh. By updating at least one fluid property of the fluid cell, the fluid-flow simulation accounts for influences due to the boundary-layer flow conditions. In this way, fluid properties (e.g., inviscid fluid property and boundary-layer fluid property) can be passed between inviscid and boundary-layer CFD simulation modules without requiring interpolation or smoothing.

Depending on the size of the fluid cells (coarseness of the fluid-flow mesh) and the curvature of the wing surface, the simulation may become choppy or stepped and, thus, in some cases, smoothing may be used to refine the results. However, the smoothing is less error inducing than as described for the techniques above without a one-to-one correlation between inviscid fluid cells and boundary-layer fluid cells.

The exemplary exchange between the inviscid CFD simulation module 1010 and the boundary-layer CFD module 1050 shown in FIG. 10 is merely an illustrative example. In some cases, modules other than the inviscid CFD simulation module 1010 and the boundary-layer CFD module 1050 perform one or more of the operations described above. In particular, determining the intersecting fluid cell and operation 1052 for determining the boundary-layer prediction point may be performed by modules other than the CFD simulation module 1010 and the boundary-layer CFD module 1050.

Depending on the particular simulation, the process described above may be iterated several times until a steady-state solution is reached. Thus, the data exchange between the CFD simulation module 1010 and the boundary-layer CFD module 1050 may occur multiple times as the simulation is iterated.

4. Computer and Computer Network System

The embodiments described herein are typically implemented as computer software (computer-executable instructions) executed on a processor of a computer system. FIG. 11 depicts an exemplary computer system 1100 configured to perform any one of the above-described processes. Computer system 1100 may include the following hardware components: processor 1102, data input devices (e.g., keyboard, mouse, keypad) 1104, data output devices (e.g., network connection, data cable) 1106, and a user display (e.g., display monitor) 1108. The computer system also includes non-transitory memory components including random access memory (RAM) 1110, hard drive storage 1112, and other computer-readable storage media 1114.

Processor 1102 is a computer processor capable of receiving and executing computer-executable instructions for performing any of the processes described above. Computer system 1100 may include more than one processor for performing the processes. The computer-executable instructions may be stored on one or more types of non-transitory storage media including RAM 1110, hard drive storage 1112, or other computer-readable storage media 1114. Other computer-readable storage media 1114 include, for example, CD-ROM, DVD, magnetic tape storage, magnetic disk storage, solid-state storage, and the like.

FIG. 12 depicts an exemplary computer network for distributing the processes described above to multiple computers at remote locations. One or more servers 1210 may be used to perform portions of the processes described above. For example, one or more servers 1210 may store and execute computer-executable instructions for receiving information for generating a computer-generated simulation. The one or more servers 1210 are specially adapted computer systems that are able receive input from multiple users in accordance with a web-based interface. The one or more servers 1210 are able to communicate directly with one another using a computer network 1220 including a local area network (LAN) or a wide area network (WAN), such as the Internet.

One or more client computer systems 1240 provide an interface to one or more system users. The client computer systems 1240 are capable of communicating with the one or more servers 1210 over the computer network 1220. In some embodiments, the client computer systems 1240 are capable of running a web browser that interfaces with a web-enabled system running on one or more server machines 1210. The web browser is used for accepting input data from the user and presenting a display to the user in accordance with the exemplary user interface described above. The client computer 1240 includes a computer monitor or other display device for presenting information to the user. Typically, the client computer 1240 is a computer system in accordance with the computer system 1100 depicted in FIG. 11.

Although the invention has been described in considerable detail with reference to certain embodiments thereof, other embodiments are possible, as will be understood by those skilled in the art.

We claim:

1. A computer-implemented method of generating a fluid-flow simulation over a computer-generated aircraft surface, the computer-generated aircraft surface comprising of a surface mesh of surface mesh polygons, the method comprising: obtaining a fluid-flow mesh for simulating a fluid flow over the aircraft surface, the fluid-flow mesh comprising a plurality of fluid cells; determining at least one inviscid fluid property for each of the fluid cells using an inviscid fluid simulation that does not simulate fluid viscous effects; identifying a set of intersecting fluid cells of the plurality of fluid cells that intersects the aircraft surface; identifying at least one surface mesh polygon of the surface mesh for at least one intersecting fluid cell of the set of intersecting fluid cells, wherein a number of the identified surface mesh polygons is fewer than a total number of surface mesh polygons that are intersected by the at least one intersecting fluid cell; determining a boundary-layer prediction point for each identified surface mesh polygon; determining at least one boundary-layer fluid property for each boundary-layer prediction point using the at least one inviscid fluid property of the corresponding intersecting fluid cell and a boundary-layer simulation that simulates fluid viscous effects; and determining at least one updated fluid property for at least one fluid cell of the plurality of fluid cells using the at least one boundary-layer fluid property and the inviscid fluid simulation.

2. The computer-implemented method of claim 1, wherein identifying the at least one surface mesh polygon for the at least one intersecting fluid cell comprises:
obtaining a centroid of the at least one intersecting fluid cell of the set of fluid cells;
identifying a surface mesh polygon having a centroid that is closest to the centroid of the at least one intersecting fluid cell;

3. The computer-implemented method of claim 2, wherein the centroid of the at least one intersecting fluid cell is the centroid of a region of the at least one intersecting fluid cell that is outside of the aircraft surface.

4. The computer-implemented method of claim 1, wherein the at least one boundary-layer fluid property includes a boundary-layer thickness value, the boundary-layer thickness value representing a distance from the aircraft surface where fluid viscous effects can be ignored.

5. The computer-implemented method of claim 1, wherein the at least one boundary-layer fluid property includes a transpiration flux value, the transpiration flux value representing a direction and an amount of fluid flow originating from the aircraft surface.

6. The computer-implemented method of claim 1, wherein the fluid-flow mesh is constructed using a Cartesian mesh, wherein the Cartesian mesh comprises:
   a plurality of flow cells, each flow cell bounded by six flat faces where opposite faces are parallel and adjacent faces are orthogonal.

7. The computer-implemented method of claim 1, wherein the inviscid fluid simulation is an Euler-based flow simulation.

8. The computer-implemented method of claim 1, wherein the at least one inviscid fluid property includes a fluid velocity vector, a fluid density value, and a fluid pressure value.

9. The computer-implemented method of claim 1, wherein the computer-generated aircraft surface is an airplane wing.

10. A computer-implemented method of generating a fluid-flow mesh for simulating a fluid flow over a computer-generated aircraft surface, the computer-generated aircraft surface comprising a surface mesh of surface mesh polygons, the method comprising instructions for:
    obtaining a fluid-flow mesh for simulating a fluid flow over the aircraft surface, the fluid-flow mesh comprising a plurality of fluid cells;
    determining at least one inviscid fluid property for each of the fluid cells using an inviscid fluid simulation that does not simulate fluid viscous effects; and
    determining at least one boundary-layer fluid property for each of the fluid cells using a boundary-layer simulation that simulates fluid viscous effects; and
    determining at least one updated flow property for at least one fluid cell using an updated fluid simulation that simulates fluid viscous effects;

11. A non-transitory computer-readable storage medium comprising computer-executable instructions for generating a fluid-flow simulation over a computer-generated aircraft surface, the computer-generated aircraft surface comprising of a surface mesh of surface mesh polygons, the instructions comprising instructions for:
    obtaining a fluid-flow mesh for simulating a fluid flow over the aircraft surface, the fluid-flow mesh comprising a plurality of fluid cells,
obtaining a fluid-flow mesh for simulating a fluid flow over the aircraft surface, the fluid-flow mesh comprising a plurality of fluid cells;

determining at least one inviscid fluid property for each of the fluid cells using an inviscid fluid simulation that does not simulate fluid viscous effects;

identifying an intersecting fluid cell of the plurality of fluid cells that intersects the aircraft surface;

determining a single boundary-layer prediction point for the intersecting fluid cell resulting in a one-to-one correlation between the intersecting fluid cell and the boundary-layer prediction point;

determining at least one boundary-layer fluid property for each boundary-layer prediction point using the at least one inviscid fluid property of the corresponding intersecting fluid cell and a boundary-layer simulation that simulates fluid viscous effects; and

determining at least one updated fluid property for at least one fluid cell of the plurality of fluid cells using the at least one boundary-layer fluid property and the inviscid fluid simulation.