Adaptive Refinement based Integral Method for Load Transfer in Fluid/Structure Interaction Problems

Pan Cheng¹, Zekai Ding² and Wu Zhang²,*

¹School of Computer Science and Engineer, Shanghai University, 99 Shangda Road, Shanghai, 200444, P.R. China
²Shanghai Institute of Applied Mathematics and Mechanics, Shanghai University, 99 Shangda Road, Shanghai, 200444, P.R. China
*wzhang@shu.edu.cn

Abstract. In fluid-structure interaction simulations using a partitioned approach, especially aeroelastic analysis, each physics component solves on its own mesh, and the interface between these meshes are in general non-matching. Force and displacement must be transferred from one mesh to another. Generally speaking, two classes of methods have been developed and applied in engineering: point wise interpolation and mapping method. For practical problems, point wise interpolation is not good enough in conversation and mapping method requires detailed meshes to describe surfaces of both fluid side and structure side. This paper presents a flexible, conservative and robust method which only requires surface mesh on fluid side. The merits of new method in terms of conversation properties were discussed. Also, the numerical results of practical problems are demonstrated in this paper.

1. Introduction
For simulations of complex system that require integration of multiple physical disciplines, such as fluid dynamics and structural mechanics, two approaches were adopted to solve this kind of problems. First is the monolithic approach which couples the governing equation of both fluid dynamics and structural mechanics, discretizes both parts as a single computational domain and solves on a whole mesh. The monolithic approach allows exact time synchronization but requires a new solver and mesh[1]. Moreover, the associated matrices of the whole strong-coupling system is likely to be ill-conditioned. However, these disadvantages are addressed by partitioned approach which allows to use the existing solver on both sides. Although partitioned approach cleverly circumvented the development of a new solver, it raises a difficult question in terms of data transfer between fluid mesh and structure mesh.

In general, fluid side is solved by computational fluid dynamics (CFD) which requires relatively fine spatial discretization, while structure is normally modelled as finite element model or even simpler as beams and then solved by stiffness equation. Since fluid and structure are solved on different meshes due to difference in characteristics, it is necessary to pass the aerodynamic loads from the aerodynamic surface to the structural grids consistently.

Data transfer is an indispensable subroutine in fluid structure interaction (FSI) problems. For a typical FSI problem like static aeroelastic analysis, one should first obtain the force distribution on the surface mesh that represents wing model through computational fluid dynamics (CFD). Then, the force distributions must be transferred consistently to the structure model so that the deflection may be
computed through computational structure dynamics (CSD). Similarly, the deflections then need to be transferred to the surface mesh even to the volume mesh. For existing methods that dealing with data transfer in FSI problems, it can be generally divided into two main categories: point wise method and mapping method. Point wise methods only require arbitrary point clouds of both aerodynamic mesh and structural mesh[2] so that point wise method is easy to be realized. Although the total loads can be transferred conservatively through point wise method, it is short for transferring the distribution of quantities from the source mesh to the target mesh[3]. Mapping methods not only require data on points, but also connectivity information on the interface. The topology information on both sides help to improve conservation and minimize error during data transfer. But in practical problems, structure part is likely to lack description of surface so that mapping methods are not able to deal with certain kind of problems.

The purpose of this paper is to present an innovative load transfer method that is capable of transferring data from CFD surface mesh to incomplete finite element(FE) model without surface description. The basic idea of this method is to rebuild surface mesh for scattered FE grids by adaptive refinement of existed CFD surface mesh. With a rebuild mesh, the concentrated loads on scattered grids are calculated by integral. The newly created surface mesh is an approximation of Voronoi diagram. This load transfer method is easy to be implemented and parallelized, and it is also able to keep balance between accuracy and efficiency.

2. Problem Statement and Summarization
The FSI problem is solved by CFD and CSD alternatively and exchange data on the boundary $\Gamma$ between each other. On structure side, the elastic equation is formulated in Lagrangian coordinates in which grids is moving with structure and boundary is denoted as $\Gamma_s$. On fluid side, the Navier-Stokes/Euler equation is solved in Eulerian coordinates in which meshes are fixed and fluxes are calculated in those volumes cells. The boundary of fluid side is denoted as $\Gamma_f$. The pressure and viscous stress are calculated by CFD and stored on the surface mesh of fluid side. The loads on the grids on structure side need to be transferred from the known pressure on the boundary mesh. The total force and moment should be conservative and more than that, the distributive force and moment should be matched before and after transferring. Usually, the fluid side is called source and structure is called target.

Mapping method includes many typical algorithms such as mortar element methods[4] and common refinement method[5][6]. This class of methods have strong mathematical foundation and conservation requirement are proved to be strictly satisfied. The quantities on source and target side are treated as function $f$ and $g$ separately. The two functions are denoted by the linear combination of the shape functions and quantities on $n$ nodes on target and $m$ nodes on source, which are $f = \sum_{i=1}^{m} f_i \phi_i$ and $g = \sum_{i=1}^{n} g_i \psi_i$. The error of load should be minimized during transfer so that the $L_2$ norm of $g - f$ should be minimized. The requirement leads to a linear system $Mx = b$, in which $M_{ij} = \int_{\Omega} \psi_i \psi_j \, dx$ and $b_i = \int_{\Omega} \psi_i f \, dx$. The discretization of $L_2$ minimization is the key to calculate integral on right and left side of the equation above[3].

Different discretization are available, including source-based, target-based and common refinement. The first two classes calculate $\int_{\Omega} \psi_i f \, dx$ by integrate over source mesh or target mesh. The common refinement is different from above two, it utilizes the overlap of meshes on both side and integrate over the smallest subdomains created by intersection. The accuracy, stability and efficiency are proved to be best among most discretization schemes since the common refinement fully use the geometry information on the surface mesh. However, this feature also limits the application of common refinement because the surface mesh on both side are necessary and should be based on the exactly same geometry.

Point wise method, also called interpolation or mesh free methods, is based on radial basis function (RBF)[7], multivariate spline[8] or any type of multivariate interpolation method[9][10]. These methods are used to interpolate displacement of the points on the fluid structure boundary and the
concentrated force are transferred by virtual work conservation over the boundary. Discretization of displacement on \( F \) is formulated as \( \delta \mathbf{u}_f = H \delta \mathbf{u}_s \), where the \( H \) is the interpolation matrix. The virtual work performed by the fluid loads and the structure stress should be equal

\[
\delta W = \delta \mathbf{u}_f^T \cdot \mathbf{F}_f = \delta \mathbf{u}_s^T \cdot \mathbf{F}_s
\]

where \( \mathbf{F}_f \) is the discretized concentrated force on fluid points and \( \mathbf{F}_s \) are the same one on structure grids. Then the force relation can be deduced as

\[
\mathbf{F}_f = H^T \cdot \mathbf{F}_s
\]

In different interpolation methods, the \( H \) matrix can be constructed based on variant basis functions. The earliest interpolation in aeroelastic is proposed in[11], in which thin plate surface(TPS) function are used as the basis function to transfer loads. TPS function is the fundamental solution of infinite plate equations and a special case of RBF, which minimizing the bending energy of thin plate and has the form \( \phi(||x||) = ||x||^2 \log(||x||) \). Then many interpolation based on different RBFs are proposed, such as multiquadric, Gaussian or Wendland \( C^n \) functions. Some of them are compact supported, which means the affect range ( circle in 2D and sphere in 3D) of these basis function is adjustable to balance between accuracy and efficiency[12]. Besides RBFs, some other methods use kriging interpolation or BEM to construct \( H \) matrix.

The interpolation method is easy and flexible to use since the equation is simple and only points are used. However, interpolation also have some drawbacks in practical problems. The first is that the distribution of quantities is not strictly conservative because it uses a different basis function instead of the source or target basis function in mapping methods. The basis functions used in mapping implies the boundary geometry information while the basis functions used in point wise method have no relevant to the geometry. It is not big deal when the points are dense and smooth enough but some time it is hard to achieve and big error will come out. Second, a \( m \times m \) linear equations need to be solved to gain \( H \) matrix where \( m \) is the grids number on structure side. When \( m \) is very large, it cost to much time to solve a large linear equation which is not worthwhile in FSI coupling process. Although reduce method are introduced[13], the cost is to lower the accuracy. Third, in some interpolation such as RBFs based, three or more noncolinear points must be specified while it is not satisfied when the structure is a beam like model and auxiliary point should be added.

3. Adaptive refinement based method

In engineering environment, the mapping and point wise method are not satisfied in some special cases mentioned above. A better method should be designed to avoid the drawbacks listed above and a list of requirements are introduced as follow.

1. The loads received on structure side should be point wised. This means that the surface mesh on structure side will not be necessary, only points are needed to transfer loads.

2. The conservation should be satisfied, say \( \int \Omega |f - g|^2 \, d\Omega \) and \( \int \Omega |f - g| \, d\Omega \) should be minimized or close to minimization, which means the loads should be conservative before and after transfer.

3. The computation should be robust and efficiency enough to be adaptive to most situation encountered in engineering.

Consider discretized \( E_f \) is represented by discrete point set \( P_f \), while \( E_f \) is also discretized as points set \( P_f \) and cells set \( C_f \). On structure side, only points are given while on fluid side, both points and cells are given. The boundary is shown as figure 1.

The basic ideal of the new method is similar to common refinement method, but only surface mesh on fluid side are utilized. There should be a surface mesh on structure side so that to perform intersection, however it is not existed on structure side and it should be reconstructed by existed surface mesh on fluid side. A simple way to reconstruct the surface mesh on structure side is to apply Voronoi diagram[14]. In Voronoi diagram, the space is decomposed to polygon cells and every cell
match to one of given points. Arbitrary point on the edges of these cells has the same distance to two nearest points in given set. The Voronoi diagram of $\Gamma^s_1$ in figure 1 is shown as (a) in figure 2. This tessellation treats every structure points as equality based on Euclidian distance.

![Figure 1. The boundary of structure and fluid side and their discretization](image1)

![Figure 2. The Voronoi diagram of structure points and overlap of two meshes.](image2)

After the surface mesh reconstructed by Voronoi diagram, the common refinement scheme could be applied by intersecting two meshes theoretically. However, generation of Voronoi diagram on three dimensional surface could be very difficult and integration operation need to be performed after surface mesh generation and intersection. A natural thought is to combine these two steps together, so the adaptive refinement on surface mesh on fluid side is introduced. The edge of Voronoi diagram can be approximated by splitting and the edge will be displayed as zigzag curve along the small cells generated by adaptive refinement. The implementation of the procedure will be described as following.

Let’s consider a case of only one fluid cell. For an arbitrary cell $c_i \in \mathcal{C}_f$ on fluid mesh, a geometry center of $c_i$ is defined as $\hat{c}_i$ and its two nearest neighbour points $p^k_{c_i} | k = 1, 2$ can be found in $\mathcal{P}_s$ if the distance

$$d\left(p^k_{c_i}, \hat{c}_i\right) = \min_k \left(d(p \mid p \in \mathcal{P}_s, \hat{c}_i)\right)$$

where distance is defined as Euclidean distance. Let’s denote $d\left(p^k_{c_i}, \hat{c}_i\right)$ as $d^k_{c_i}$. The edge of Voronoi diagram of this cell should be the intersection of the middle plane of $p^k_{c_i} | k = 1, 2$ and the cell. The intersection line on the cell could be approximated by recursively splitting the cell. First, we define a measurement of the cell size as $s_i$ for cell, for example the longest edge of triangular cell or longest diagonal of quadrilateral cell. If $s_i > |d^1_{c_i} - d^2_{c_i}|$, the cell should be split into smaller sub cells and we classify the pieces by distance. Then, for an arbitrary structure point $p_j \in \mathcal{P}_s$, the nearest cells(including split and non-split cells) will be found through the process above and we put them into a subset denoted as $C^j_{sub}$. Recursively perform the process above until the measurement of the cell size
smaller than a predefined threshold $\varepsilon$. The Voronoi cell of $p_j$ is approximately covered by the sub cells in $C_{sub}^j$.

A sample case with one square cell with square side $l$ and three structure points in 2D space is shown in figure 3. The three structure points $p_1$, $p_2$ and $p_3$ are marked by color blue, green and red. Respectively, their sub cell sets are marked by the same colors. At the beginning, the fluid cell split into 4 pieces and their color shows which structure point they belong to. But the distance threshold didn’t satisfied, so keep splitting until the threshold reached. After 12 time splitting, the size of smallest cell is less than

![Figure 3. A square cell with 3 structure points sample case](image)

threshold $\varepsilon = l/4096$ and the splitting is stopped. The square cell are cut into 3 blocks and every block containing split cells can be regarded as approximation of Voronoi cells. The edge of the Voronoi diagram is not straight lines, but zigzags curve with small steps. Because the splitting termination condition is $\varepsilon > s_i > |d_{ij}^1 - d_{ij}^2|$ and $|d_{ij}^1 - d_{ij}^2|$ always greater than 0, $\lim_{e \to 0} |d_{ij}^1 - d_{ij}^2| = 0$. This guarantee the geometry center of the splitting cells on edge will be on the Voronoi edges when $\varepsilon$ approaches 0.

In a full fluid mesh, splitting should be performed on each cells until threshold reached, then the approximated Voronoi diagram for the whole surface will be generated. Because the splitting operation on one cell is irrelevant to other cells, it is easy to parallelize the procedure. The splitting procedure is also available on other type of mesh or even hybrid mesh. For triangular cells, the splitting cells can be generated by connect the middle point of each edge. For polygon cells, triangular cells will be generated first by connect each vertex and the geometry center then split these triangular.

After splitting procedure, the loads on structure point will be naturally calculated by integration. Suppose we have $m$ fluid cells in $C_f$ and $n$ structure points in $P_s$. Each cell in $C_f$ with pressure vector $\bar{\sigma}_i$ and viscous stress vector $\bar{\tau}_i$ will be split into $m_i$ sub cells. The $j$th sub cell in these $m_i$ sub cells have area of $a_{ij}$ and the vector from reference point to its geometry center defined as $r_{ij}$. Normally fluid side is solved by finite volume method, the pressure and viscous stress is averaged, so the values on sub cells are the same to their parent cell. The total loads from the fluid side on boundary, including force and moment is discretized as

$$F_f = \int_\Omega f \, dx = \sum_{l=1}^m \sum_{j=1}^{m_i} (\bar{\sigma}_{ij} + \bar{\tau}_{ij}) \, a_{ij}$$

$$M_f = \int_\Omega f \times r \, dx = \sum_{l=1}^m \sum_{j=1}^{m_i} (\bar{\sigma}_{ij} + \bar{\tau}_{ij}) \, a_{ij} \times r_{ij}$$

While on the structure side, for each structure point $p_l$, an approximated Voronoi cell combined by split fluid cell is created and contain $n_l$ sub cells.

$$F_s = \int_\Omega f \, dx = \sum_{l=1}^n \sum_{j=1}^{n_l} (\bar{\sigma}_{ij} + \bar{\tau}_{ij}) \, a_{ij}$$

$$M_s = \int_\Omega f \times r \, dx = \sum_{l=1}^n \sum_{j=1}^{n_l} (\bar{\sigma}_{ij} + \bar{\tau}_{ij}) \, a_{ij} \times r_{ij}$$
Since $\sum_{l=1}^{n} n_i = \sum_{m=1}^{m} m_l$ and every sub cell are covered in integration, the integral of force and moment on both side are the same. Thus, the conservation of total force and moment $F_f = F_s$ and $M_f = M_s$ are satisfied. The conservation of distribution is not strictly satisfied due to the grain size of the minimum split cells. We will show more detail in next section.

### 4. Numerical results of Common Research Model

We have implemented the adaptive refinement based method into a code called AECS. In this section, we will apply load transfer method to NASA Common Research Model (CRM) to test the consistency of quantities before and after transferring CRM was developed by NASA’s Subsonic Fixed Wing (SFW) Aerodynamic Technical Work Group. This wing-body-horizontal (without nacelle-pylons) configuration is representative of a contemporary high-performance transonic transport detailed description[15].

![Fluid surface mesh](image1) ![Finite element model](image2)

**Figure 4.** Coupling surface of NASA Common Research Model

Figure 4 shows the outer wing region of the CFD and CSM coupling surface of NASA CRM model. The wing surface is discretized by 11522 quadrilateral elements and 12099 vertices. The structure part of CRM wing is represented by equivalent wing-box model which is provided by the official site of CRM from NASA and is discretized by 4626 nodes, 8476 quadrilateral elements and 14 triangular elements. Since the element distribution on CFD surface mesh and wing-box model differ a lot, point-wise method is expected to achieve poor results in transferring distribution of quantities from source mesh to target mesh. For static aeroelastic analysis, researchers tend to pay more attention to moment distribution along certain direction instead of force at a specific point so that they could obtain the static balance position of the wing. In this section, we will compare the moment distribution along span-wise direction between point wise method and adaptive refinement based method. Also the influence of threshold value on the error of moment distribution between surface mesh and structural mesh will be compared.

Firstly, we started from a RANS CFD simulation, which is computed on the fluid mesh. Our test case description is shown in table 1.

| Parameter          | Value |
|--------------------|-------|
| Mach Number        | 0.85  |
| Angle of Attack    | 2.5°  |
| Reynolds Number    | $5 \times 10^6$ |
| Reference Temperature | 100°F  |

Table 1. Description of CRM test case.

Then, we transferred the pressure distribution on fluid mesh to structural mesh using two different kind of methods. One is TPS interpolation, which is an RBF interpolation using $\phi(||x||) = ||x||^2 \log(||x||)$. But in practical problems, researchers tend to divide fluid mesh into several groups so
that the force on fluid mesh could be transferred group by group to the corresponding structural grids using TPS interpolation and will greatly improve the conservation of force distribution before and after transferring. To compare the results of grouped TPS interpolation and original TPS interpolation, both interpolation methods were adopted. The other method is adaptive refinement based method. To count the moment distribution along specific direction, we divided the wing into 19 sections by 20 planes, construct a local coordinates and accumulate the moment distributed from tip to certain plane along span-wise direction. The divided CFD surface mesh is shown in figure 5. Each color represents a section divided by planes. GID is the ID of section and varies from 0 to 18.

![Figure 5. Sections divided by planes.](image)

Since our method determines whether to split the fluid cell or not by calculating the threshold $\epsilon$, we set $\epsilon = 0.1$ which is approximately $1/10$ of the longer side of the largest cell in surface mesh and start splitting the surface mesh. Figure 6 shows the grid at the wing tip before and after splitting. The surface mesh is totally refined by 929583 cells and 1542594 points based on wing-box model so that the aerodynamic loads could be transferred conservatively.

![Figure 6. Fluid mesh of wing tip before and after splitting ($\epsilon = 0.1$)](image)

After splitting the surface mesh on fluid side, the software starts to transfer the aerodynamic loads on surface mesh to wing-box model and calculates the accumulated moment along the direction we gave. The moment distribution before and after transferring are shown in the figure 7. We could find out that the moment distribution on fluid mesh differs a lot from that transferred to structure grids by the original TPS interpolation. Grouped TPS interpolation improves conservation significantly comparing to the original TPS interpolation but is still inferior to the adaptive refinement based method. In general, the adaptive refinement based method is the most conservative method comparing to the point wise methods.

When the threshold value $\epsilon$ is rather large, none of the cell on fluid mesh need to be split. Then the adaptive refinement method degenerates to nearest neighbour search method. To determine how threshold value $\epsilon$ affects the error of moment distribution, we split the surface by two threshold value
0.1, 0.5 and compare the absolute error of moment distribution with the surface mesh without splitting. The absolute error of moment distribution is defined as $Error = |M_f - M_s|$ in which $M_f$ represents the moment on fluid surface mesh and $M_s$ represents the moment that transferred to structure. Results in figure 8 indicate that as the threshold $\varepsilon$ decreases, the error of moment distribution is also minimized especially at the section around $X/L = 0.2$. In general, a relatively small threshold value helps to decrease the error of moment distribution and to some extent average the error along the direction we provided.

![Figure 7. Moment distribution along span-wise direction (solid line: moment distribution before transferring, circle: adaptive refinement based method, triangle: grouped TPS interpolation, square: TPS interpolation)](image)

From the results above, we can draw out a conclusion that our method reconstructs the topology of surface mesh, enrich information in fluid cells and generate a new surface mesh that is suitable for a more conservative data transformation based on structural grids.
5. Conclusions
In this paper, we overviewed a family of load transfer methods in fluid-structure interaction problems and presented a flexible, conservative method that only requires surface mesh on fluid side and grids on structure side to transfer force from fluid to structure. This method can be utilized as a subroutine in fluid-structure interaction problems especially the aeroelastic problems. When dealing with practical problems that fluid and structure do not share the same geometrical support such as ‘fishbone’ and ‘wing-box’ model, point-wise method can be inaccurate in transferring quantity distribution. The method we presented in this paper combines the advantages of point wise methods and mapping methods. By reconstructing the topology of surface mesh based on structural grids, the error will be minimized and the distribution of quantities will be transferred consistently from source mesh to target mesh comparing to the point wise method. Moreover, surface mesh on structure side is not required which is likely to lack in practical problems. As a result, the adaptive refinement based method can be implemented more extensively than mapping methods. In addition, the threshold value can be adjusted to determine whether to pursue efficiency or accuracy. Finally, our claims were validated by numerical results of Common Research Model.

References
[1] Hüblner B, Walhorn E and Dinkler D 2004 A monolithic approach to fluid-structure interaction using space-time finite elements Comput. Methods Appl. Mech. Eng. 193 2087–2104
[2] Rendall T C S and Allen C B 2008 Unified fluid – structure interpolation and mesh motion using radial basis functions Int. J. Numer. Meth. Engng 74 1519–1559
[3] Jiao X and Heath M T 2004 Common-refinement-based data transfer between non-matching meshes in multiphysics simulations Int. J. Numer. Meth. Engng 61 2402–2427
[4] Farhat C, Lesoinne M and Letallec P 1998 Load and motion transfer algorithms for fluid / structure interaction problems with non-matching discrete interfaces: Momentum and energy
conservation, optimal discretization and application to aeroelasticity *Comput. Methods Appl. Mech. Engrg.* **157** 95-114

[5] Jiao X and Heath M T 2004 Overlaying Surface Meshes, Part I: Algorithms *Int. J. Comput. Geom. Appl.* **14** 379–402

[6] Jiao X and Heath M T 2004 Overlaying Surface Meshes, Part II: Topology Preservation and Feature Matching *Int. J. Comput. Geom. Appl.* **14** 403–419

[7] M. D. Buhmann 1993 New developments in the theory of radial basis function interpolation *Multivar Approx. From CAGD to Wavelets*, K. Jetter Fl Utreras, eds World Sci. pp 35–75

[8] Smith M J, Hodges D H and S. Cesnik C E 2008 Evaluation of Computational Algorithms Suitable for Fluid-Structure Interactions *J. Aircr.* **37** 282–294

[9] Dai K Y, Liu G R, Lim K M and Gu Y T 2003 Comparison between the radial point interpolation and the Kriging interpolation used in meshfree methods *Comput. Mech.* **32** 60–70

[10] Smith M J, Hodges D H and S. Cesnik C E 2008 Evaluation of Computational Algorithms Suitable for Fluid-Structure Interactions *J. Aircr.* **37** 282–294

[11] Harder R L and Desmarais R N 1972 Interpolation using surface splines *J. Aircr.* **9** 869–871

[12] Boer A De, Schoot M S Van Der and Bijl H 2007 Mesh deformation based on radial basis function interpolation *Comput. Struct.* **85** 784–795

[13] Rendall T C S and Allen C B 2009 Efficient mesh motion using radial basis functions with data reduction algorithms *J. Comput. Phys.* **228** 6231–6249

[14] Aurenhammer T 1991 Voronoi diagrams *ACM Comput. Surv.* **23** p 94

[15] Vassberg J 2011 A Unified Baseline Grid about the Common Research Model Wing/Body for the Fifth AIAA CFD Drag Prediction Workshop 29th AIAA Applied Aerodynamics Conference 2011