Wind Turbine Blade FSI (Part 1) - Mesh

Authors: Sebastien Lachance-Barrett (Cornell University) & Edwin Corona (University of Waterloo)

Problem Specification
1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Numerical Solution
6. Numerical Results
7. Verification & Validation

Mesh

Named Selections

We start off by naming various faces of our geometry for later use in FLUENT and to make surface body referencing much easier when creating our mesh.

Summary of steps in the above video:

1. Suppress Blade FEA
2. Show the fluid geometry, wireframe.
3. Create named selections
    a. Inlet
    b. Inlet-Top
    c. Outlet
    d. Blade (select all then control click the surfaces not wanted)
    e. Periodic 1 (one of the quadrilateral surfaces).
    f. Periodic 2 (the other quadrilateral surface).
    g. Fluid

IMPORTANT

If you are using ANSYS 16 or later while doing this project, please name the quadrilateral surfaces Period 1 and Period 2 instead Periodic 1 and Periodic 2. In versions 16 and later of ANSYS, the word periodic is a key word used for something that is not relevant to this project and using this name will cause problems later on.

When you enter the FLUENT interface, if your "period 1" and "period 2" named selections are not appearing, try renaming them "surface 1" and "surface 2".

Default Mesh and Section Plane

The section plane is a really neat tool that allows you to cut through the mesh in order to visualize its interior!

Global Mesh Controls

We start by applying some global mesh settings which means that these settings will be applied to the whole mesh altogether.

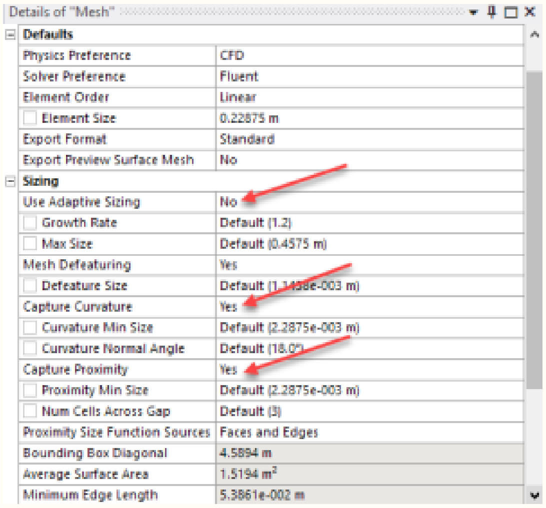
Note: There have been changes to the Mesher since this video was recorded. Please see the notes under the video.

Summary of steps in the above video:

1. Automatically optimized for CFD and FLUENT, tetrahedral cells.
2. Change use advanced size function to proximity and curvature. It makes it deal with curves better, lower skewness.
3. Change relevance center to medium.

Version Changes (ANSYS 19.1 or later)

Skip the relevance center settings shown in this video. Your "Details of Mesh" window should look like the image below:


Details of "Mesh"

Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
Element Size	0.22875 m
Export Format	Standard
Export Preview Surface Mesh	No
Sizing	
Use Adaptive Sizing	No
Growth Rate	Default (1.2)
Max Size	Default (0.4575 m)
Mesh Defeaturing	Yes
Defeature Size	Default (1.1438e-003 m)
Capture Curvature	Yes
Curvature Min Size	Default (2.2875e-003 m)
Curvature Normal Angle	Default (18.0°)
Capture Proximity	Yes
Proximity Min Size	Default (2.2875e-003 m)
Num Cells Across Gap	Default (3)
Proximity Size Function Sources	Faces and Edges
Bounding Box Diagonal	4.5894 m
Average Surface Area	1.5194 m^2
Minimum Edge Length	5.3861e-002 m

Local Mesh Controls

After applying controls to the whole mesh, we now apply mesh settings to specific areas of our geometry.

Summary of steps in the above video:

1. Insert Match Control
    a. Select the two trapezoid faces for the high and low geometry selection
    b. Choose axis of rotation to be global coordinate
    c. Match control is for the nodes to match up for the periodic sides.
2. Insert face sizing
    a. Use the name selection, blade surface.
    b. Input sizing of 0.3m.
    c. Behavior should be hard.
3. Insert local inflation around the blade
    a. Geometry is the whole body
    b. For boundary, select named selection and choose the blade (click enter)
    c. Keep default settings for the rest
4. Add a sphere of influence
    a. First, create a new coordinate system.
        i. Define it by named selections and choose the blade surface.
    b. Insert body sizing
    c. Geometry, the whole body
    d. Type: sphere of influence
    e. Sphere center: Coordinate System
    f. Radius: 30m
    g. Element size: 2m
5. Click Generate

ANSYS 19.1 Users:

If your mesh fails when you insert a sphere of influence, skip this step. Instead, change the growth rate to 1.15 from the default of 1.2. To do this, highlight Mesh in the tree, then look under Sizing.

Mesh Metrics

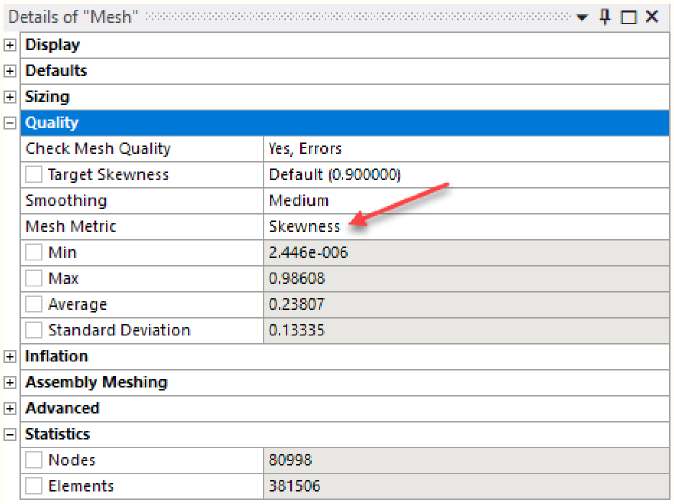
We now show you how to judge the quality of your mesh. This is an extremely important step because a bad mesh can lead to bad results!

Summary of steps in the above video:

1. Skewness and orthogonal quality are the two most important. Skewness better for CFD.
2. Look at the skewness mesh metric under details of mesh.
3. Click on different ranges to see the specific cells in that range

Version Changes (ANSYS 18.2 or later)

In newer versions, the Mesh Metrics can be viewed under Quality instead of under Statistics:



It is generally advised to keep the minimum orthogonality greater than 0.15 and maximum skewness lower than 0.95. Having bad cells or elements can lead to incorrect simulation results. However, these are general guide rules and depend on the physics solved or where the cells are located. The following tables can help you gauge the quality of your mesh.

Skewness:

Outstanding	Very Good	Good	Sufficient	Bad	Inappropriate
0-0.25	0.25-0.50	0.50-0.80	0.80-0.95	0.95-0.98	0.98-1.00

Orthogonal quality:

Inappropriate	Bad	Sufficient	Good	Very Good	Outstanding
0-0.001	0.001-0.15	0.15-0.20	0.20-0.70	0.70-0.95	0.95-1.00

Go to Step 4: Physics Setup

Go to all FLUENT Learning Modules