WIND FORCE METHODS: PROJECTED AREA, PROJECTED PRESSURE, VELOCITY COMPONENT

1- Question : January 3, 2009

I have been trying to understand the three wind force methods (Projected Area (PA), Projected Pressure (PP) and Velocity Component (VC)) discussed in API 2008 4F3rd Edition's Annex, Sections B.8.3.3. Even after all this review I feel I still need a better understanding.

To help me understand how SAFI applies theses methods we have made some simple models with Tubing Members or Surfaces. I may have made some modeling mistakes but the results from some of these simple SAFI models did not seems to match my understanding of API's intent in their Sections 8.3.3, 8.3.3.2, 8.3.3.6 & B.8.3.3.

The models made of Surfaces only, where we can select Projected Area or Velocity Component, seem to give results which follow my understanding of API 2008. FYI we did use as shape factor of 1.5 on these Surface Models to match the factor used on tubing.

However when we use Tubing Members and allow SAFI to pick the shape factor (1.5) and the method to calculate wind forces to results do not seem to match the Velocity Component Method API suggest for a bare structure & Windwalls (8.3.3 & B.8.3.3). Instead the results seem to match the Projected Area Method which API suggests for Appurtenances other than Windwalls (Ref API SectionB.8.3.3, bottom of page 38 to 39).

I base my conclusions on the X & Z reaction summary generated by SAFI vs. hand calculations using the API formulas from Sections 8.3.3 & B.8.3.3.

Answer

The results of SAFI are correct according to our interpretation. Let me explain our interpretation. First take a look at the figure of the clause 8.3.3.5. The area of the member A = w*L, where L is the total length of the member and *w* the *projected width* of member. So in the code the wind is applied perpendicularly to that projected surface, no matter which shape is the section. The results would be the same for any shape of section. In this code the velocity approach is more related to the inclination of the member according to the wind. Please have a look at Ki factor ($(\sin \varphi)^2$) the clause 8.3.3 and the figure of the clause 8.3.3.2. You can see than in this figure the resultant load is applied perpendicularly to the surface *A*. Since the displayed section is rectangular, the resultant direction of the load would not be as displayed if the load would be applied to each faces of the section separately. Also, please note that in the figure B.8.3.3, the horizontal dark line represent the length of the member (not the width of the member).

In conclusion, for a vertical member the reaction of load will always be in the same direction of the wind even with the velocity approach. For an inclined member, then the reaction should always be perpendicular to the surface A (=w*L) of the member. For a vertical surface, depending on the approach as illustrated in your example, the reaction direction may differ to the wind.

Try this same example for a vertically inclined member (0,0,0) to (12,12,0) ft with a wind applied in the X direction. Then you will see the effect of the wind velocity component method. You can also do the example with a horizontally inclined member (0,0,0) to (12,0,12) ft with a wind applied in the X direction. The conclusion should be the same.

API WIND WALL, WARNING

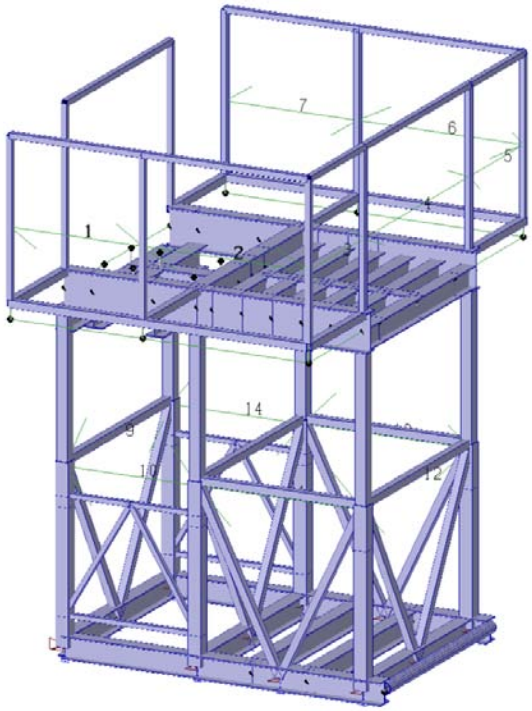
2- Question : August 18, 2009

The attached model contains my first use of the new API Wind Wall feature. I have defined two wind walls (Upper& Lower) each with an X, Z & 296.6 Deg. wind applied.

The lower wind wall (#1) seems to apply the loads per the API Table 8.6, except I keep getting what I will call a caution note about "The Wind Wall 1" when I open up the Surface load tables. However the upper wind wall (#2) does not apply the loads per API but acts like individual load surfaces.

The basic loads where I apply the wind wall loads are:
X direction in Load Case #8
Z direction in Load Case #10
296.6 deg. direction in Load Case #13

Can you please help me on the following?
1) What does this *Caution Note* about "The Wind Wall 1" mean, I do not understand it?
2) What am I doing wrong with the upper wind wall and how can I change it to make it perform per API?



Answer

Please note that temporarily, if you get a warning when opening the table of load on surfaces related to the wind walls, you should orient the normal of the surfaces of opposite's sides with the same direction (not as outward or inward normal for all surfaces). This warning does not affect results for 4 sided wind walls wind forces, but will affect results for 3 sided wind walls wind forces. This issue will be fixed on next update. So remember the internal axis "Z" of the surfaces should point in the same global orientation (This is due to a minor bug which has been fixed).

Please note that even if your warning is displayed only for "Wind wall 1", it is because we display only the first warning. If you fix this wind wall you will see that the warning will also be displayed for the other wind wall. This is the reason why you get wrong Cs for "Wind wall 2".

So to fix the problem in your model you need to use the "Revert connectivity" button in the bottom right part of the "Edit Surface" (Short cut is "U") command on half of the surfaces of the wind walls for both wind walls. The internal axis "Z" of the surfaces is displayed when you activate the "Edit surface" command, so it should be easy to fix. Please note that by doing that you will also need to change the "Distribute load on sides ..." option to still distribute loads on the correct side of the surfaces.

We did compare results after doing the modification in your model; all the warnings disappear when opening the "Wind load on Surface" table. Also, the "Cs" values are correct after this modification.

API WIND LOAD, PLATE, PROFILE

3- Question : February 26, 2009

1) We have used plate elements in a Mast model. When I was adding API wind loading on the structure I noticed that plate elements did not have the ability to use the API Wind Profiles to define the plate loading. Have I missed something or is this option not yet in SAFI? Is there any easy way to let SAFI apply the API profile to plates?

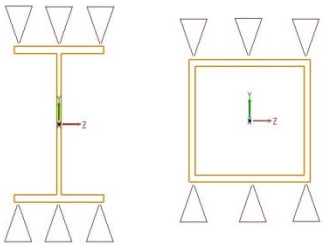
2) In Member Attributes under the "Steel Tab" there is the ability to change the Major and/or Minor Axis *K* & *L* values. However it uses the terms *Kx*, *Lx* for the Major Axis & *Ky*, *Ly* for the Minor Axis. If one uses the "Delete a member" curser it defines the member's axis for column buckling as Y & Z. So how does one determine the Major & Minor Axis orientation of a given member, especially for square sections?

Answer

1) Currently, the API wind loads are available on members and surfaces but not on plate elements. At this moment, you will need to either use a surface loading area or calculate the wind pressure and apply to the plate elements.

2) *Kx*, *Lx* and *Ky*, *Ly* are named based on the convention of the design code. The strong axis corresponds to the Y member axis (e.g. *Kx and Lx* are the parameters for flexural buckling where the rotation is around the *Z* axis).

If there are supports that prevent the displacements of the member along its *Y* axis, the support prevents the member to buckle on the strong axis. These supports are controlled by *Kx*, *Lx*. On the other hand, if there are supports that prevent the displacements of the member along its Z axis, the supports prevent the member to buckle on the weak axis. These supports are controlled by *Ky*, *Ly*.



On the image above, the supports prevent buckling on the strong axis (prevent the rotation around the Z axis or said differently, they prevent the displacements along the Y axis). These supports are controlled by Kx, Lx parameters.