Chapter 3: Methods of Analysis

3.1 Assumptions

During the course of this research, many assumptions were made which have a significant impact on the results that follow in later chapters. The major assumptions are discussed in detail below.

3.1.1 End of analysis

For each analysis the external loads are increased on the structure until one of the following situations occurs:

1) A bifurcation point is reached on the load-frequency graph
2) A limit point is reached on the load-frequency curve
3) The deflections or load magnitudes become excessive
4) The analysis stops because the incremental load step is too small. This is characteristic of local buckling on the arches.

If the analysis is stopped due to a bifurcation or limit point instability, the final load is referred to as the buckling load. If the analysis stops due to the solution diverging, then the failure is due to local buckling or wrinkling. For other cases, the analysis is stopped at a load which is larger than what can reasonably be expected on the structure.

3.1.2 Material

The arch material is assumed to be a linearly elastic, isotropic material with a modulus of elasticity of 7 GPa (1015 ksi) and Poisson’s ratio of 0.3. Analyses of this type often neglect the
self-weight of the structure; however, the structure under consideration is very large. Because of its size, the self-weight of the structure may be significant; therefore, it is included in the analysis. A density of 1440 kg/m$^3$ (90 lb/ft$^3$) is used. These values are representative of a lightweight woven fabric such as Kevlar or nylon.

3.1.3 Nonlinearity

The structural system that is modeled in this research is expected to be very flexible because of its size, material, and small thickness relative to the radius of the cross section. Therefore, the model is expected to exhibit large displacements. In order to account for the large displacements, geometric nonlinearity is considered. The geometrically nonlinear solution method takes into account the displaced shape of each element when formulating each successive load increment. Because the solution is history dependent, the time increments must be kept small. This is to ensure that the strains and rotations within each increment are sufficiently small, so that the history dependent effects are modeled correctly.

Since there is a possibility of unstable behavior during the analysis, the modified Riks method is used during the application of external loads. The modified Riks method includes the load magnitude as an additional unknown in the problem formulation. This approach gives solutions regardless of whether the response is stable or unstable. The benefit of the modified Riks method is that the equilibrium states during an unstable phase of the load-deflection response can be found. For this research, post-buckling equilibrium states are not desired; however, the response at the buckling load is of critical importance. Therefore, it is required that the response is accurately modeled up to and including the point of instability.
3.1.4 Contact

For the case of the leaning arches, the two arches are initially in contact at the crown of the arches. During the load application, the contact between the arches is expected to change; therefore, the surfaces of the arches must be modeled as contact regions. Since it is initially unknown what the contact area between the arches will be or if the arches will come into contact at points away from the crown, each arch was defined as a surface. Each surface is then used to model the contact between the arches. One surface was defined as the master surface and the other defined as the slave surface. For three-dimensional contact between shells, the small sliding option in ABAQUS is required. For this option, the master surface is not smoothed; therefore, ABAQUS assumes that a slave node which comes into contact with a master surface slides along a line formed by the two adjacent nodes on that surface. This is computationally more efficient than finite sliding, which allows the slave nodes to slide over the actual master surface. However, it must be insured that the slave node does not move far from its original point of contact.

3.2 Bifurcation versus limit point buckling

Instability of structures can occur in two ways: the first is bifurcation buckling and the second is limit point buckling. Bifurcation buckling is an instability in which there is a sudden change of shape of the structure. A bifurcation point is a point in a load-deflection space where two equilibrium paths intersect. On a load-frequency curve, a bifurcation point can be characterized by the load-frequency curve passing through a frequency of zero with a non-zero slope.

Limit point buckling is an instability in which the load-displacement curve reaches a maximum and then exhibits negative stiffness and releases strain energy. During limit point buckling there are no sudden changes in the equilibrium path; however, if load is continuously increased then the structure may jump or “snap” to another point on the load-deflection curve. For this reason, this type of instability is often called “snap-through” buckling, because the structure snaps to a new equilibrium position. A limit point is characterized by the load-frequency curve passing
through a frequency of zero with a zero slope. The load-deflection curve also has a zero slope at the point of maximum load (limit point).

3.3 Leaning arches

The structure being considered in this thesis consists of two arches that lean against one another. The typical arches are shown in Figure 3.1. The function used to define each arch in a vertical plane is:

\[ z(x) = 17 - \frac{17}{12.5} x^2 \]  

(3.1)

where \( z \) and \( x \) are in meters.

The two arches are secured at the point where they come together; therefore, the displacements and rotations are constrained to be the same at that point for each arch. Two tilt angles (\( \theta \)) are considered, 15° and 30°. A hollow cross section is considered with a radius of 0.4 m (1.31 ft) and a shell thickness of 2.5 mm (0.098 in.).

Figure 3.1 Geometry of leaning arches
3.4 Boundary conditions

The boundary conditions of the actual structure are unknown. One option for modeling the bases of the structure would be to use springs or guy wires to stiffen the base. The second option, which is used in this thesis, uses both pinned and fixed bases in order to provide an upper and lower bound on the solution.

The finite element model for the base consists of a circular disk with two rings of shell elements. At the center of each base is a single node where the inner ring of elements comes together. Figure 3.2 shows the finite element model of the base.

![Figure 3.2 Base of arch](image)

The fixed base is created by restraining all six degrees of freedom at each node around the outer ring of nodes on the base. This prohibits any movement of the base of the arch as well as any rotations of the base. The pinned base restrains just the three translational degrees of freedom at the center node of each base; therefore, the base allows rotation in any direction. The stiffness of the base was increased so that the center node could be used as the pin. The modulus of elasticity of the base is 700 GPa (101,500 ksi) and the thickness of the base is 5 mm (0.197 in.).
3.5 Loads

The loads on the structure are applied in two phases. The first phase applies the self-weight of the structure and the internal pressure. For all analyses in this thesis, the internal pressure is 500 kPa (72.5 psi). The second phase applies the external loads using nonlinear analysis. Three different external loads were studied.

The first external load is a “full” snow load. This load is a vertical load that is approximately uniform. The load is applied on the arch for slopes between $-30^\circ$ and $30^\circ$ from the horizontal as concentrated loads at nodes. It is assumed that for slopes greater than $30^\circ$ along the arch, snow will fall off the tent membrane; therefore, no load will be transferred from the membrane to the arches. Around the cross section, the loads are distributed between slopes of $-45^\circ$ and $45^\circ$ from the vertical, regardless of the tilt angle. The distribution of loads along the arch and around the cross section is shown in Figure 3.3. This assumes that the tent transfers load only to the top of the cross section, where the tent fabric is in contact with the arches. For the parabolic arch being studied, given by (3.1), the loads are applied between $x = -2.653$ and $x = 2.653$ m (8.70 ft). This distribution results in the load being applied to 175 nodes for a single arch and 350 nodes for two leaning arches.

![Figure 3.3 Full snow load distribution](image-url)
The second type of external load is a “half” snow load. This is essentially the same as the full snow load, except the loads are only applied on slopes between 0° and 30°, as shown in Figure 3.4. This loading models unsymmetric loads on the structure. The application of load to the cross section is the same as for the full snow load. For the parabolic arch considered, the loads are applied between x = 0 and x = 2.653 m (8.70 ft). This distribution results in the load being applied to 91 nodes for a single arch and 182 nodes for two leaning arches.

![Figure 3.4 Half snow load distribution](image)

For the full and half snow loads, the distance between the two sets of leaning arches is treated as an unknown; therefore, load is reported as a total load in kN rather than a uniform pressure. The total load is the summation of all the concentrated forces on the nodes on the arches. If the module separation is known, then a snow pressure on the tent can be calculated. However, if a minimum design pressure must be satisfied, then the module separation can be calculated.

The final type of external loading is a wind load. This load is a non-uniform pressure that is always normal to the surface of the arches, and is a nonconservative load. For the same reasons as the snow loads, the pressure is only applied around the cross section from –45° to 45°. However, unlike the vertical loads, it is applied along the entire length of the arches. Various wind distributions will be studied further in Chapter 4. The wind loads on the structure are reported in terms of the wind pressure, which is the basic dynamic pressure, $g_w$. 
3.6 Computer analysis

This thesis uses the commercial finite element program ABAQUS (Hibbitt, et al., 1994) for the numerical analysis. The element selection, mesh generation, and convergence study are discussed in the following sections. A sample ABAQUS program is included in Appendix A.

3.6.1 Element selection

The shell element S4R is used for the analysis. The S4R element is a three-dimensional, doubly-curved, four-node shell element with six degrees of freedom per node that uses bilinear interpolation. The shell element uses reduced integration, is applicable to both thin and thick shells, and can be used for finite strain applications. The differences between the ten general shell elements available in ABAQUS are discussed below.

3.6.1.1 Thin versus thick shells

Elements S8R and S8RT are for thick shells. These elements account for the transverse shear flexibility within the shell during the analysis. Elements STRI3, STRI35, S4R5, STRI65, S8R5, and S9R5 are thin shell elements. These six elements are to be used only when the shell thickness is small and when the transverse shear flexibility can be neglected. The S3R and S4R elements can be used for either thin or thick shells. These two elements use thick shell theory as the thickness increases and therefore allow for transverse shear deformation. As the shell thickness decreases, the element becomes a discrete Kirchhoff thin shell element, and therefore the shear deformations become negligible. The S3R and S4R elements are referred to as general purpose shell elements because of their ability to act as both thin and thick shells.
3.6.1.2 Finite strain versus small strain

Elements S3R and S4R are finite strain elements. These two elements allow for large strains and account for changes in the shell’s thickness. As the strain is increased on the element, the thickness will change depending on the value of Poisson’s ratio that is specified. If Poisson’s ratio is zero, the shell thickness will not change, and if Poisson’s ratio is 0.5, then the element will behave as an incompressible material.

The other eight general shell elements are small strain elements. These elements neglect any change in thickness as the shell deforms. Since the thickness of the finite strain elements changes as load is increased, these elements calculate stress and strain based on the current deformed state. Therefore, they give true stresses and strains rather than engineering stresses and strains, which are based on the original element geometry.

3.6.1.3 Degrees of freedom

ABAQUS provides elements with five (three translations and two in-surface rotations) and six (three translations and three rotations) degrees of freedom per node. The elements with five degrees of freedom per node are STRI35, S4R5, STRI65, S8R5, and S9R5. These elements may be more economical but can not be used in finite strain applications and they can only be used for thin shells. STRI3, S3R, S4R, S8R, and S8RT have six degrees of freedom per node and are applicable to thin/thick shell or finite/small strain applications as stated in the preceding sections.

3.6.1.4 Other modeling issues

The following are some other issues associated with modeling shell elements.

1) Elements S4R, S4R5, S8R, S8R5, S9R5, S3R, and S8RT use reduced integration to form the element stiffness. Reduced integration can provide more accurate
results and reduced running time. When reduced integration is used with first order elements, hourglass control is also used.

2) The element S8R5 may give inaccurate results for buckling problems of doubly curved elements.

3) ABAQUS automatically models doubly curved surfaces by calculating the normal at each node and creating a smooth curved surface between the nodes. The algorithm that ABAQUS uses to calculate the normal to nodes can be inaccurate for a coarse mesh, therefore a fine mesh should be used for doubly curved surfaces.

3.6.1.5 Summary

Because of the unknown effect of shear deformation and because of the possibility of large strains in the analysis of the structure being considered, a finite strain element that can be used as a thin or thick shell is desired. The element S4R appears to be the best suited for this application.

3.6.2 Mesh generation

The finite element mesh used in this analysis is created using Mathematica (Wolfram, 1993). A program that can create a generic mesh for any profile, tilt angle, or number of elements is desired. A sample program is included in Appendix B.

The process governing the position of any node on the surface of a vertical arch is as follows:

\[
s_i = \frac{i \cdot s_0}{n} \quad (3.2)
\]

\[
s_0 = \int_{x_0}^{x_0} \sqrt{1 - z'(x)^2} \, dx \quad (3.3)
\]
\begin{align*}
  z_i &= z(x_i) \quad \text{(3.4)} \\
  \sin \gamma_i &= \frac{z'(x_i)}{\sqrt{1 + z'(x_i)^2}} \quad \text{(3.5)} \\
  \cos \gamma_i &= \frac{1}{\sqrt{1 + z'(x_i)^2}} \quad \text{(3.6)} \\
  \phi_j &= \frac{j}{m} 2\pi \quad \text{(3.7)} \\
  x_{ij} &= x_i + r \cdot \sin \gamma_i \cdot \cos \phi_j \quad \text{(3.8)} \\
  y_{ij} &= r \cdot \sin \phi_j \quad \text{(3.9)} \\
  z_{ij} &= z_i + r \cdot \cos \gamma_i \cdot \cos \phi_j \quad \text{(3.10)}
\end{align*}

where: $n$ is the number of divisions along the arc length of the arch  \\
$i$ is the $i^{th}$ division along the arc length of the arch ($0 \leq i \leq n$)  \\
$s_i$ is the arc length to the $i^{th}$ point on the arch  \\
$s_0$ is the arc length of the arch  \\
$z(x)$ is the function defining the shape of the arch in the vertical plane  \\
x_0 is the first positive root of $z(x)$  \\
x_i is the x coordinate of the $i^{th}$ point on the arch and is the first positive root of 
\[ \int_0^{x_i} \sqrt{1 - z'(x)^2} \, dx - s_i = 0 \]
\[ r \] is the radius of the cross section  \\
$\gamma$ is the angle between the plane of a cross section and the vertical  \\
$\phi$ is the angle around the cross section ($0 \leq \phi \leq 2\pi$)  \\
m is the number of divisions around the cross section of the arch  \\
$j$ is the $j^{th}$ division around the cross section of the arch ($0 \leq j \leq m$)  \\
x_{ij}, y_{ij}, and z_{ij} are the x, y, and z coordinates of any point on a vertical arch

An arch that is created using the equations above can be rotated by an angle $\theta$ from the vertical plane into a tilted position using the following transformations:
\[
\psi = \tan^{-1}\left(\frac{y_{ij}}{z_{ij}}\right) \tag{3.11}
\]

\[
X_{ij} = x_{ij} \tag{3.12}
\]

\[
Y_{ij} = \sqrt{y_{ij}^2 + z_{ij}^2} \cos(\psi - \theta) \tag{3.13}
\]

\[
Z_{ij} = \sqrt{y_{ij}^2 + z_{ij}^2} \sin(\psi - \theta) \tag{3.14}
\]

where: \(X_{ij}, Y_{ij},\) and \(Z_{ij}\) are the coordinates of a leaning arch
\(\psi\) is the angle between the vertical \(XZ\) plane and any point on a vertical arch
\(\theta\) is the tilt angle from the vertical

A second, leaning arch can be created by finding \(Y_{ij_{\text{max}}}\) for the tilted arch and then copying the original arch across the \(x-z\) plane at \(y=Y_{ij_{\text{max}}}\). The new \(y\)-coordinate (\(Y_{ij_{\text{new}}}\)) becomes:

\[
Y_{ij_{\text{new}}} = 2Y_{ij_{\text{max}}} - Y_{ij} \tag{3.15}
\]

### 3.6.3 Convergence study

The finite element mesh contains two variables that affect the number of elements in the model. The first variable is the number of divisions along the arc length of the arch, \(n\). The second variable is the number of divisions around the circumference of the arch, \(m\). Varying both \(n\) and \(m\) affects the accuracy of the results. The elements throughout the convergence study have aspect ratios between 1:2 and 1:1. The model used in this study is a single circular arch with a radius of 4.88 m (16 ft), a cross-sectional radius of 0.152 m (0.5 ft), and fixed bases. The material used has a shell thickness of 6.35 mm (0.25 in.), a modulus of elasticity of 6.89 GPa (1000 ksi), and Poisson’s ratio of 0.3. The arch is inflated to a pressure of 413.7 kPa (60 psi) and then loaded to a total snow load of 42.7 kN (9600 lb)
Since an analytical solution for a pressurized arch could not be found, a mesh with 9120 elements \((m = 24, n = 380)\) was chosen as a standard with which to compare other meshes. This mesh is chosen as the largest mesh studied because the model then is already very complex with almost 55,000 degrees of freedom. If a second arch is added to the model, it is feared that a numerical instability may occur. For each mesh considered, the deflections and stresses at the apex and the lowest natural frequency at various loads are compared. The results of the convergence study are shown in Table 3.1 where the percent error reported is the maximum percent error for all six quantities.

The mesh with 4800 \((m = 24, n = 200)\) elements is chosen since all deflections, stresses, and natural frequencies are less than 1% different from the results for the 9120 element mesh.

### 3.7 Pressurized Toroid

As mentioned in section 2.6, papers by Sanders and Liepins (1963) and Tielking, et al. (1971) discuss the effect of internal pressure on the cross-sectional deformation of a toroid. For purposes of comparison, a finite element model is constructed of a half-torus that is similar to the case plotted in Tielking, et al (1971).

The model used in this section is a single circular arch with a radius of 0.4575 m (18 in.), a cross-sectional radius of 0.305 m (12 in.), and fixed bases. The material used has a shell thickness of 45.8 mm (1.8 in.), a modulus of elasticity of 6.89 GPa (1000 ksi), and Poisson’s ratio of 0.3. The arch is inflated to a pressure of 207 kPa (30 psi). The arch is divided into 1200 elements.
Table 3.1. Element discretization comparison

<table>
<thead>
<tr>
<th>Number of Elements (n x m)</th>
<th>Internal Pressure (414 kPa)</th>
<th>External Load (42.7 kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Stress (MPa)</td>
<td>Deflection (cm)</td>
</tr>
<tr>
<td>1920 (120x16)</td>
<td>5.15</td>
<td>0.2824</td>
</tr>
<tr>
<td>Percent Error</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2400 (200x12)</td>
<td>5.07</td>
<td>0.2778</td>
</tr>
<tr>
<td>Percent Error</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3200 (160x20)</td>
<td>5.19</td>
<td>0.2841</td>
</tr>
<tr>
<td>Percent Error</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4160 (260x16)</td>
<td>5.15</td>
<td>0.2820</td>
</tr>
<tr>
<td>Percent Error</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4800 (200x24)</td>
<td>5.21</td>
<td>0.2851</td>
</tr>
<tr>
<td>Percent Error</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6400 (320x20)</td>
<td>5.19</td>
<td>0.2839</td>
</tr>
<tr>
<td>Percent Error</td>
<td></td>
<td></td>
</tr>
<tr>
<td>9120 (380x24)</td>
<td>5.21</td>
<td>0.2849</td>
</tr>
</tbody>
</table>
Figure 3.5 Cross-section of pressurized toroid

The finite element model gives results which are very similar to the results published by Sanders and Liepins (1963) and Tielking, et al. (1971) Figure 3.5 shows the magnified displaced cross-section of the toroidal finite element model compared to the original shape. The cross-sectional deformation of the toroid is nearly identical; however, the finite element model developed here moves upwards at the apex. This upward movement is most likely due to the fixed ends of the model.