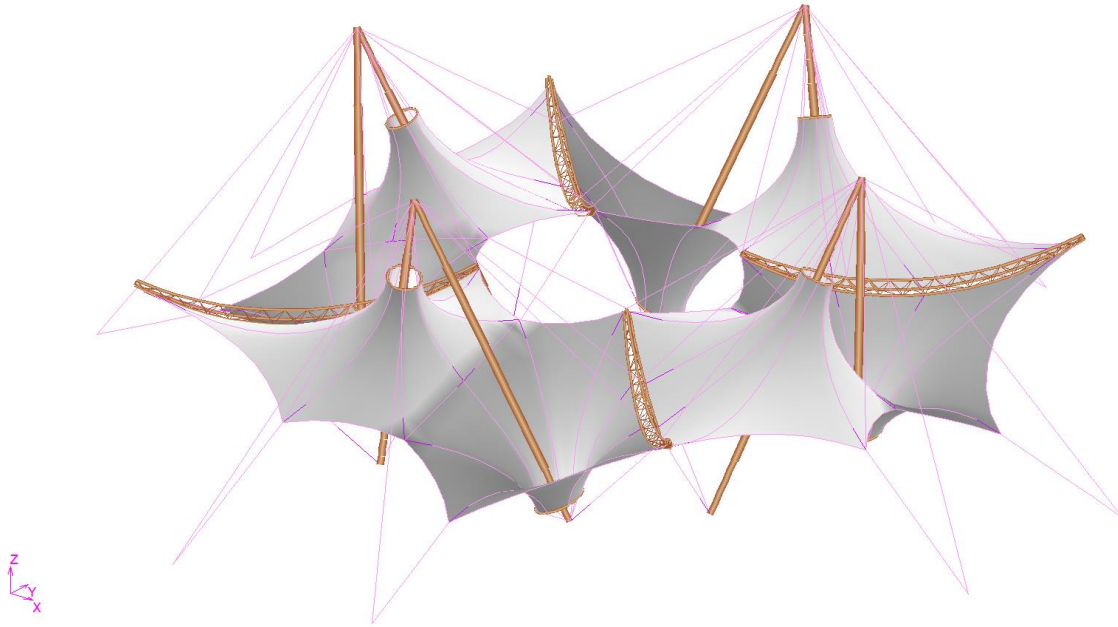# NDN Software Description

August 16th, 2023

Figure 1 – Arizona State University (ASU) Sky Song, Analysis Model

NDN is a comprehensive FEA software package developed specifically for tensile membrane engineering professionals. It provides engineers with everything they need for the design and optimization of membrane and steel structural systems. It is a single package that contains model building, analysis, post processing, member sizing, and patterning. NDN is the most comprehensive software available in the world today for the tensile membrane industry. It is easy to learn and has the fastest matrix solver in the industry. NDN automatically loads your structure, sizes the steel members, and produces patterns to any desired fabric width.

The element type library is extensive within NDN. This allows for nearly limitless modeling possibilities. There are three basic families of elements; axial force, beam, and membrane elements. Within the axial and membrane family there are several varieties bringing the total number of element types to twelve. A full description of these various element types is outlined in the Element Type Library table on page 10.

NDN is organized into eight working environments (listed below). Each environment is accessed by its corresponding "tab" located along a vertical column on the right-hand edge of the display.

- Setup
- Load

- Model
- Analysis

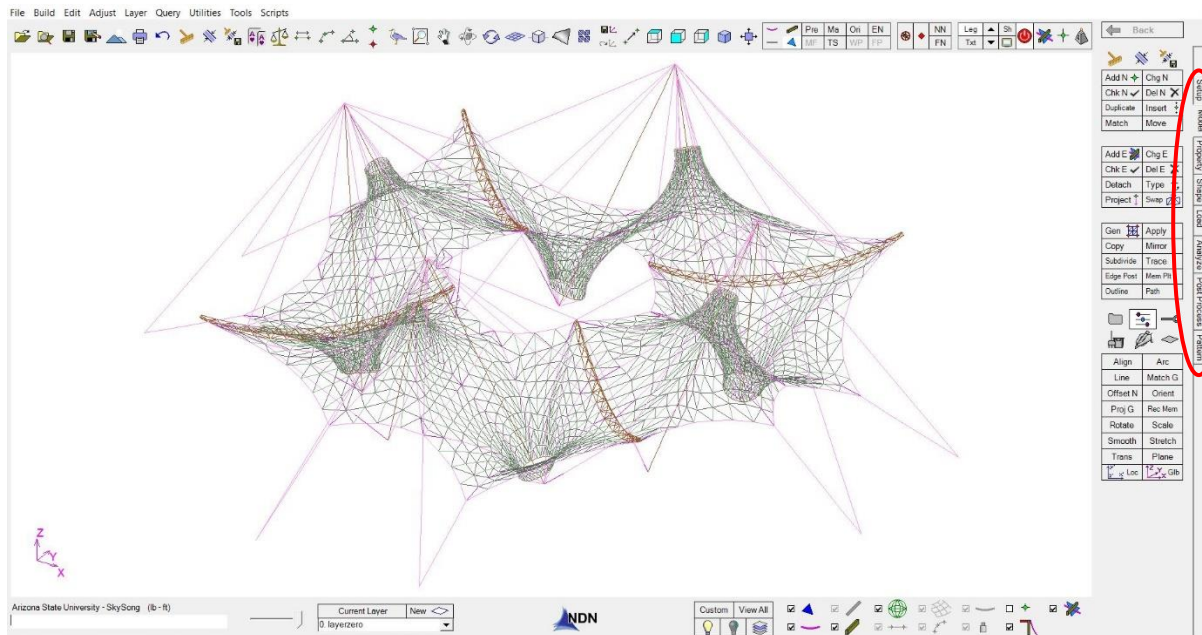- Shape
- Post Pro.

- Properties
- Pattern

Figure 2 – Screen Image, Model Environment

The display in all environments is organized in the following manner: main pull-down menus along the upper left side of the screen, shortcut buttons along the top, main control panel down the right side, and display controls along the bottom.

NDN Environments

Setup – The setup environment allows the user to define file locations, libraries, units, graphic controls, preferences, default properties, design parameters, fabric type, seam styles, etc.

Figure 3 – Graphic Control Tab of Setup Environment
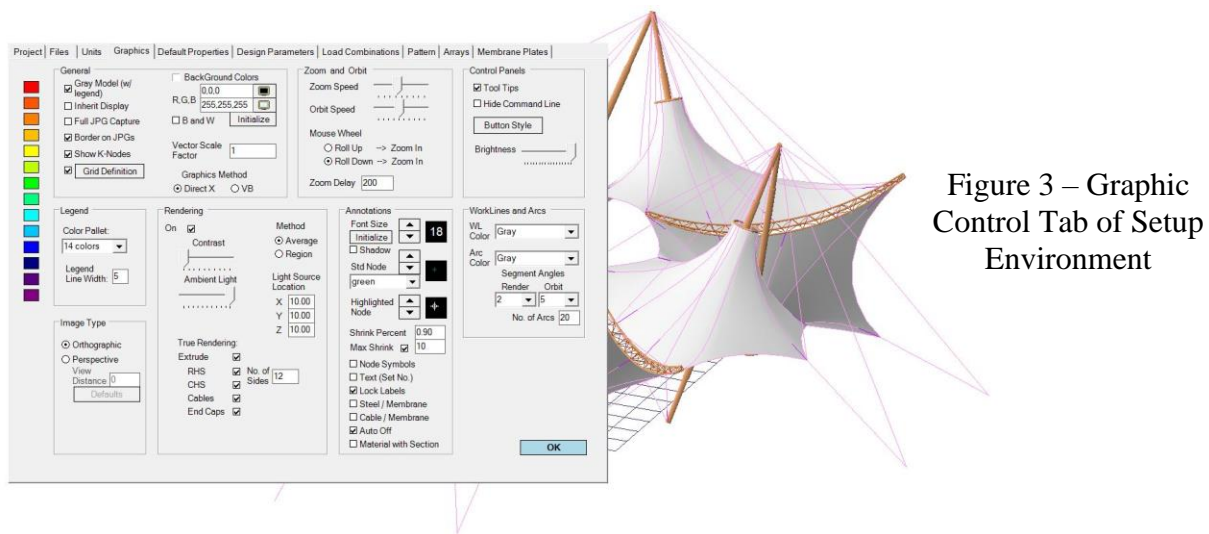
Model – The Model environment is the largest part of the system. It contains everything that is required to build and manipulate models in 3D. It is organized into eight different menu areas:

- File – For file access. Reads and writes native data files, DXF files etc.
- Build – Used to generate groups of all element types, copying, mirroring, etc.
- Edit – Used for editing models on an element by element, node by node basis.
- Adjust – Allows adjustment of the model, stretching, moving, etc.
- Layer – Defines and manages the layers, blocks, etc.
- Query – Used to provide information to the user, distances, quantities, etc.
- Utilities – Contains housekeeping algorithms, renumbering, etc.
- Tools – Provides construction tools, intersections, projections, etc.

Figure 4 – ASU Sky Song Installed. Phoenix, Arizona, United States

Shape – The shape environment is used to form find the equilibrium shape of the membrane and cable system. It operates on either the membrane components alone or on the full system as directed by the user. Also as directed by the user, it can solve for shapes controlled by a defined stress field or a defined force density. All element types can be actively and simultaneously used during shape generation. This includes beams, struts, geodesics, etc.
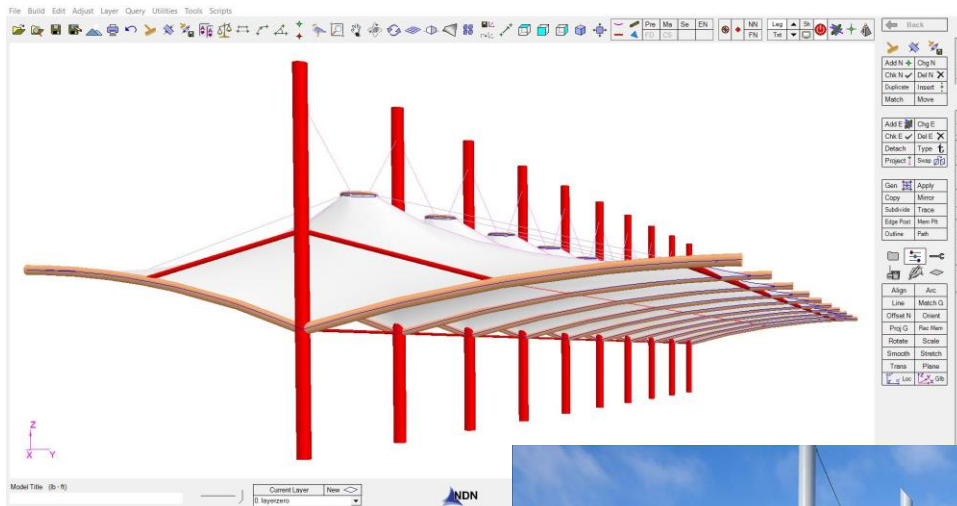

Figure 5 – Rendered Screen Image

Figure 6 – San Diego Airport, Terminal 2 Departure pavilion

Property – The property environment is used to define all the detailed properties required for full finite element analysis (FEA). Properties can be entered manually or conveniently pulled from libraries. Users can also create their own custom libraries with material and section properties unique to whatever materials their company uses most frequently.

Load – The load environment is used to generate the load cases from externally applied loads such wind, snow, and live load. It contains time saving commands and algorithms for automatic loading of models.

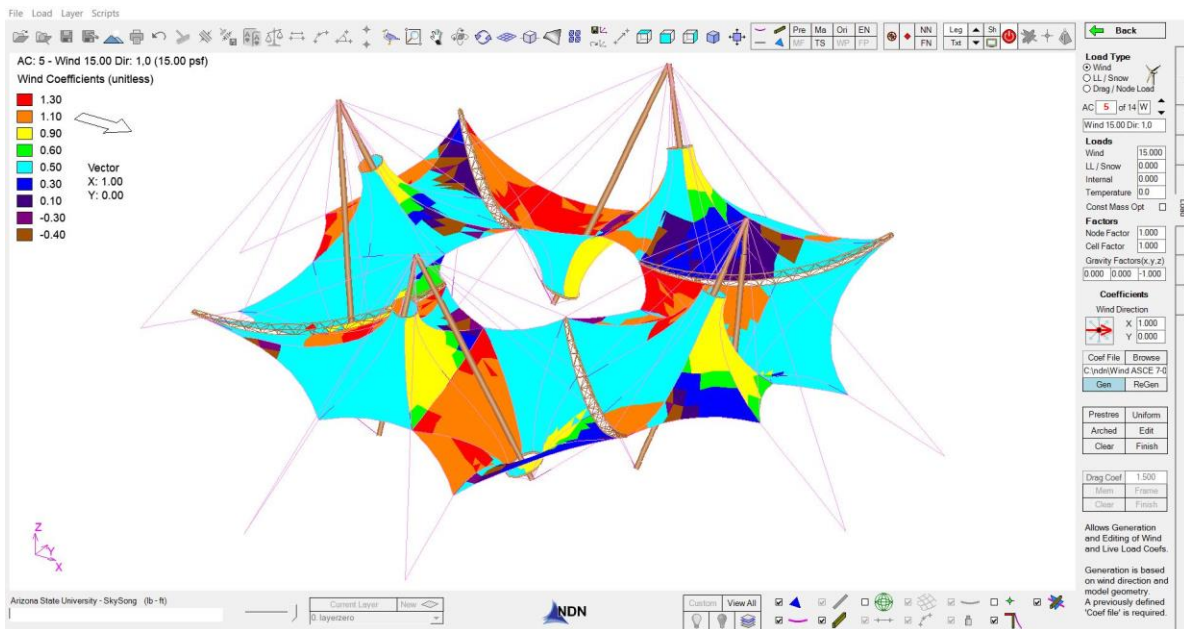Figure 6 – Wind Coefficients, Automatically Generated Based on Geometry & Building Code Requirements.

Analysis – The analysis environment performs the large deflection FEM analysis based on the stiffness method. The matrix solver is a highly optimized proprietary sparse solver. It is not limited by bandwidth and uses minimal memory. It has been tested by NASA and shown to be the fastest matrix solver available today (able to solve models with 10,000+ elements in one to two seconds). The user can execute the analysis either manually, load case by load case, or automatically run all load cases at once.
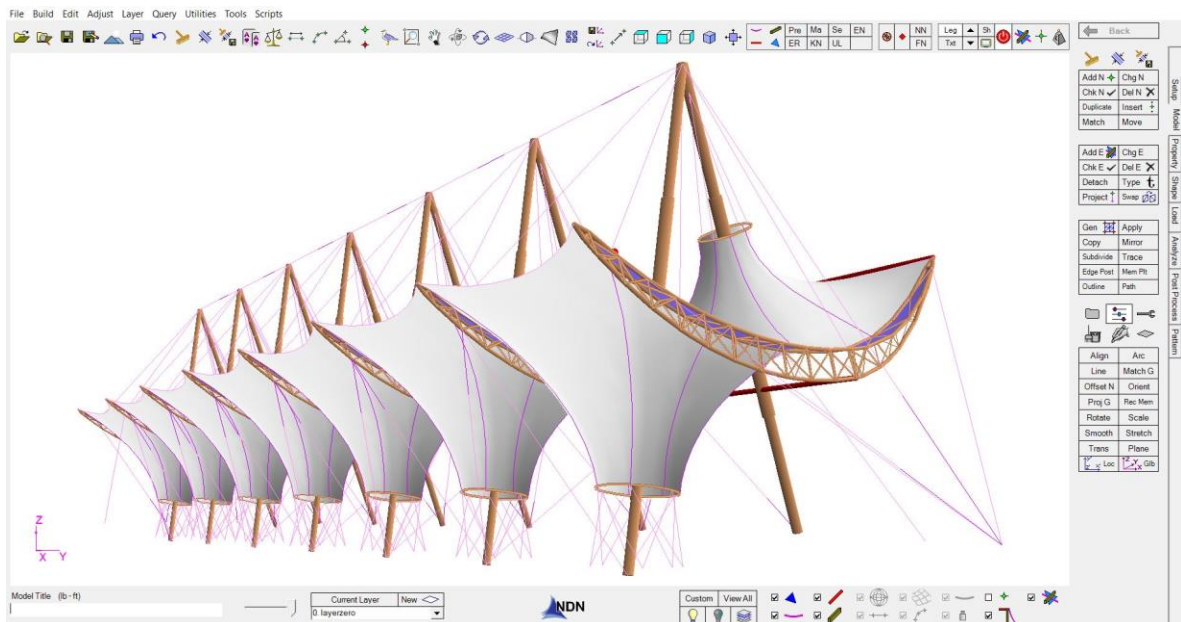

Figure 7 – Rendered Model, Rosa Parks Transit Center

5

Figure 8 – Rosa Parks Transit Center, Installed – Detroit, Michigan, USA

Post Processing – The Post Processing environment allows the user to examine and graphically display analysis results. These include deflections, member forces, bending moments, etc. It also generates custom reports to whatever level of detail is required. It performs automatic member sizing for cables, rods, circular and rectangular hollow sections using either British, EU, Australian, or US design codes. Furthermore, it performs structural checks on circular hollow connections.
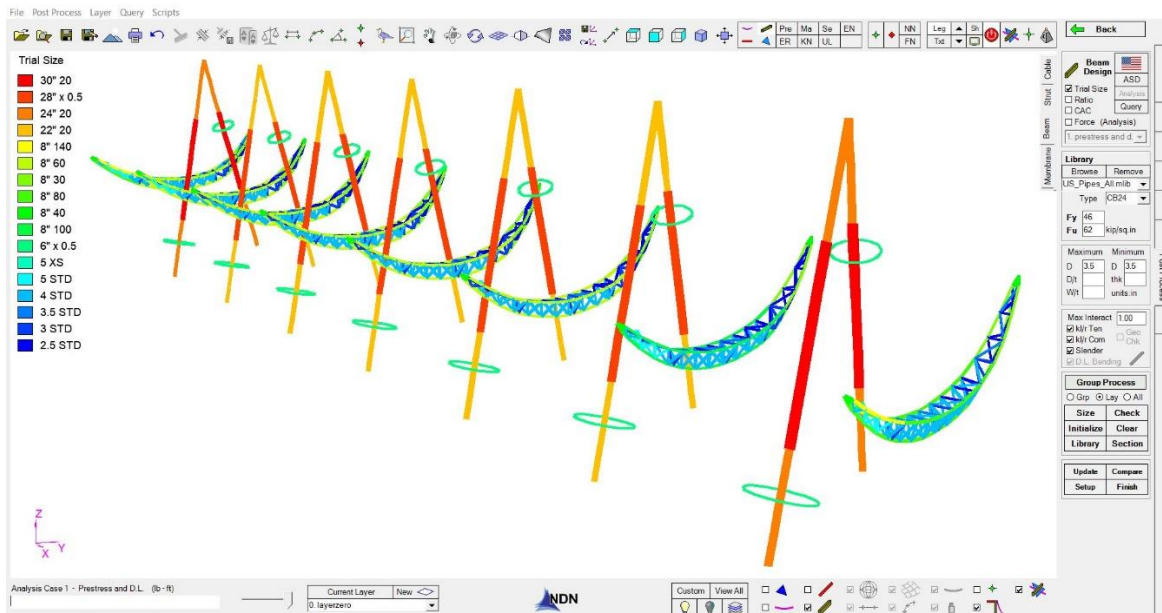

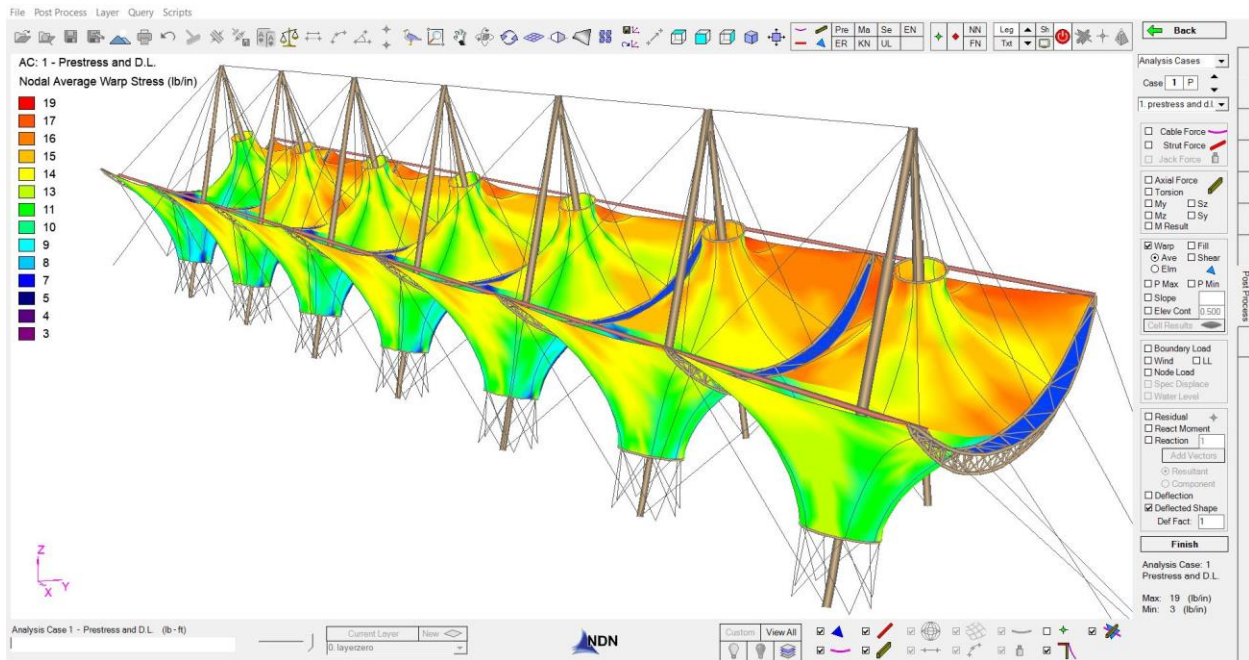Figure 9 – Beam Design Portal of Post Processor (Only Beam Elements Displayed)

Figure 10 – Membrane Stress, Post Processor

Patterning – This section preforms membrane patterning. Patterns can be defined individually by the user or generated automatically to a given fabric width. They are mathematically flattened either by geometric triangulation methods or thru highly accurate minimum energy analysis, depending on the precision required.
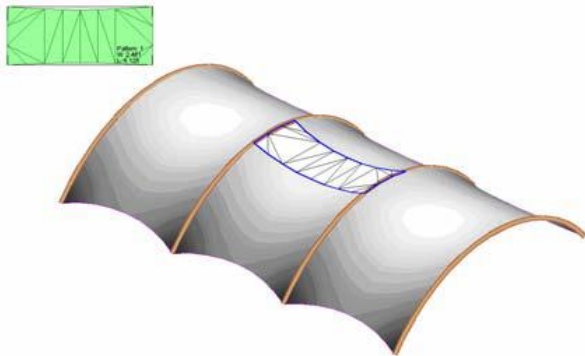

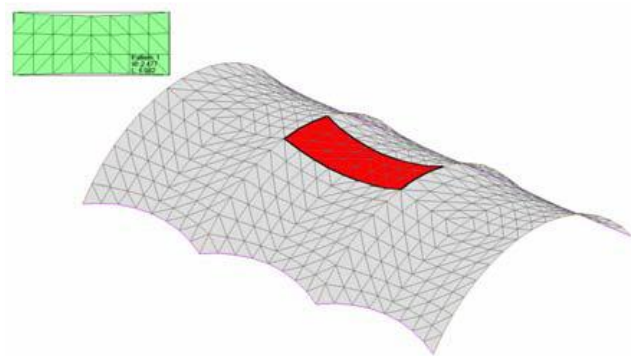Figure 11 – Pattern Flattening by
Geometric Triangulation

Figure 12 – Pattern Flattening by
Minimum Energy

Seam allowances (i.e. adds and subtracts), cut lines, etc. are automatically added to the patterns. Corners are automatically trimmed. Bolt hole marking can also be automatically transferred onto the patterns at the direction of the user. Output from the patterning module is in the form of both text files and DXF files. The output includes the original nodes and work-lines form the model, the triangulation used for flattening, seam allowances, bolt hole layouts, curve fit perimeter, numerical data, etc.
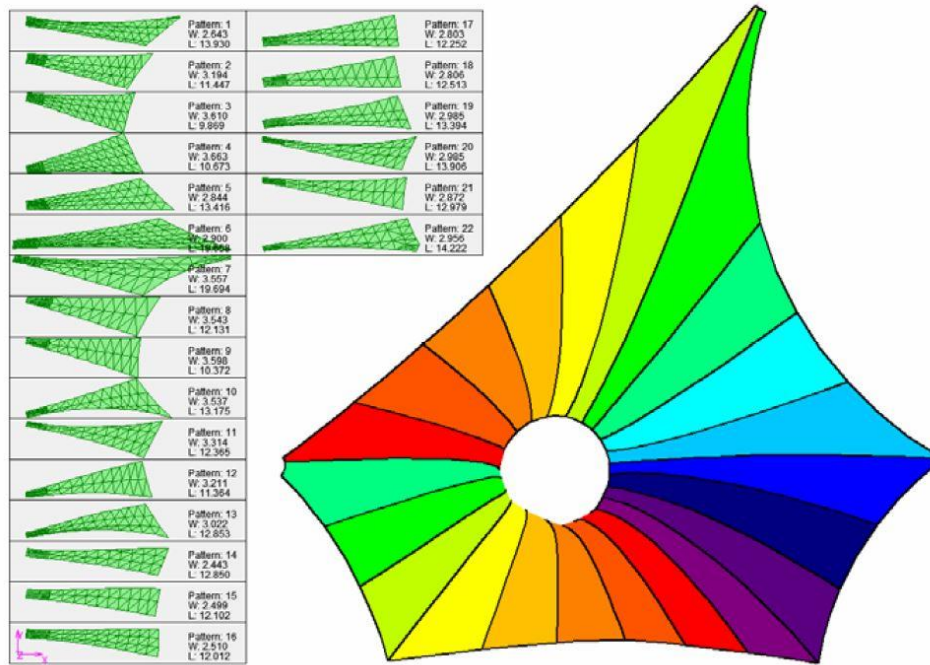
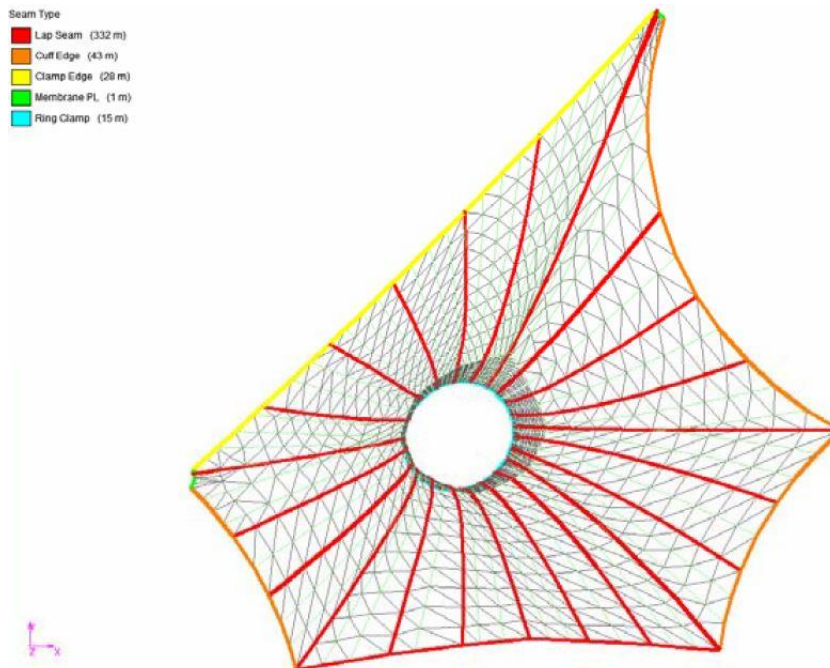Figure 13 – Screen Image, Pattern Review, Patterning Environment.

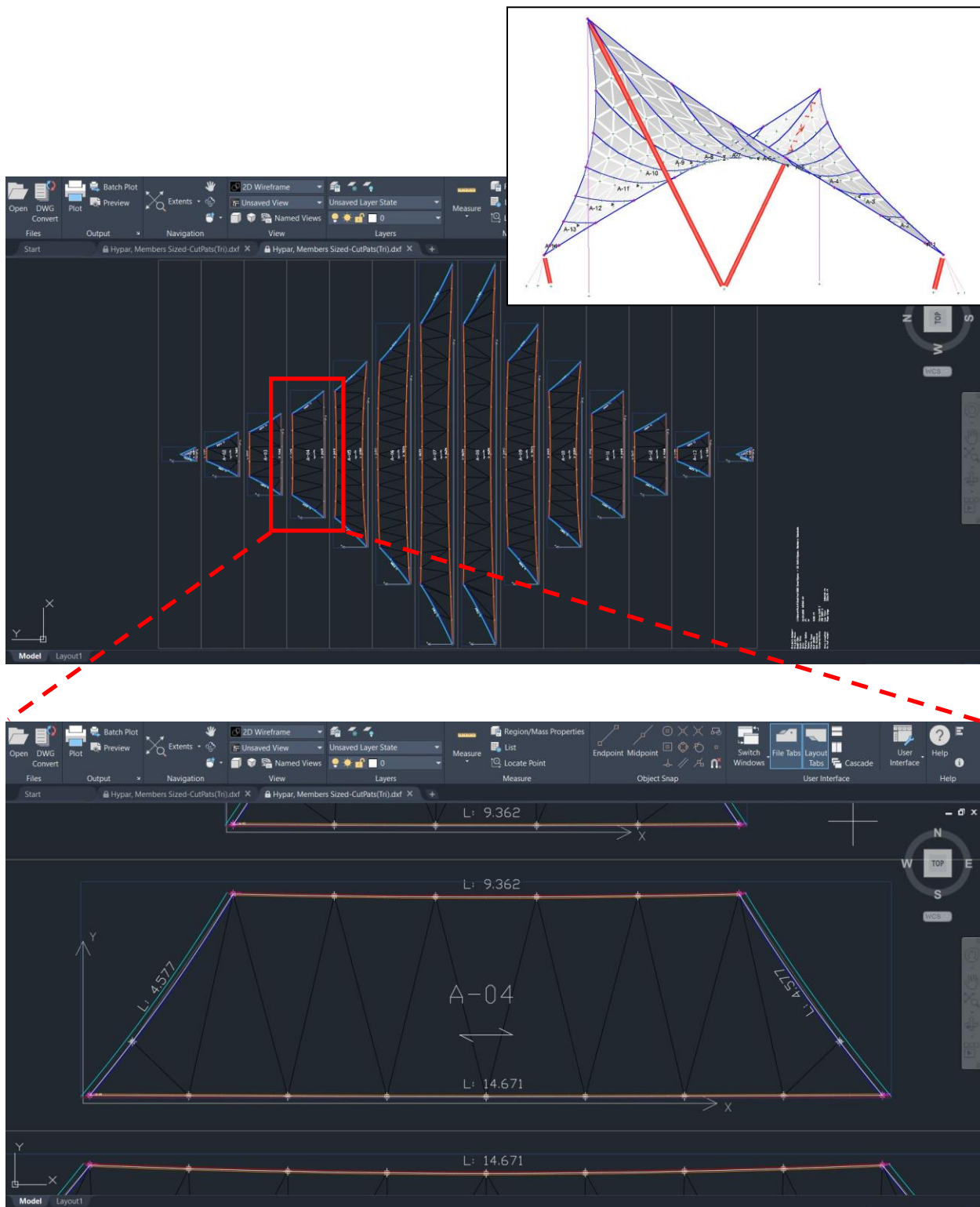Figure 14 – Seam Type, Defined on 3D Model in NDN.

Figure 15 – Patterning Output Transferred to AutoCad via. DXF File – Simple Hypar

In addition to the basic patterns and original nodes, this data also contains curve fit edges, seam allowances, match lines, bolt hole layouts, index marks, etc.

Element Type Library – The element type library is extensive, allowing almost limitless modeling possibilities. There are twelve element types in all as described below.

Element Type Library

Element Name / Type	Behavior During Shape Generation	Behavior During Analysis
Workline	Graphic Only	Graphic Only
Arc	Graphic Only	Graphic Only
Strut	Elastic - Carries Tension or Compression.	Elastic - Carries Tension or Compression.
Cable (Constant Force)	Non-elastic. Expands or contracts to maintain constant tension.	Elastic - Carries tension only.
Cable (Constant Length)	Elastic - Carries Tension Only.	Elastic - Carries tension only.
Cable (Force Density)	Non-elastic. Expands or contracts to maintain constant force density.	Elastic - Carries tension only.
Cable (Detached)	Elastic - Carries tension only.	Elastic - Carries tension only.
Jack	Expands or contracts to predefined length.	Expands or contracts to predefined length.
Geodesic	Pulls nodes into geodesic path on membrane.	Inactive
Beam	Elastic - Carries axial force, shear, moment, and torsion.	Elastic - Carries axial force, shear, moment, and torsion.
Membrane (Constant Force)	Non-elastic. Expands or contracts to maintain constant prestress.	Elastic - Carries tensile only.
Membrane (Constant Length)	Elastic - Carries tensile only.	Elastic - Carries tensile only.
Membrane (Tension/Compression)	Elastic - Carries tension or compression.	Elastic - Carries tension or compression.

Table 1 – NDN Element Types

Example Structures:

NDN is currently used by most leading tensile membrane designers, engineering consultants, and fabricators worldwide. Some A few additional structures NDN has been used for (not already highlighted above) include.

Figure 16 – BC Place Stadium Roof, Vancouver British Columbia
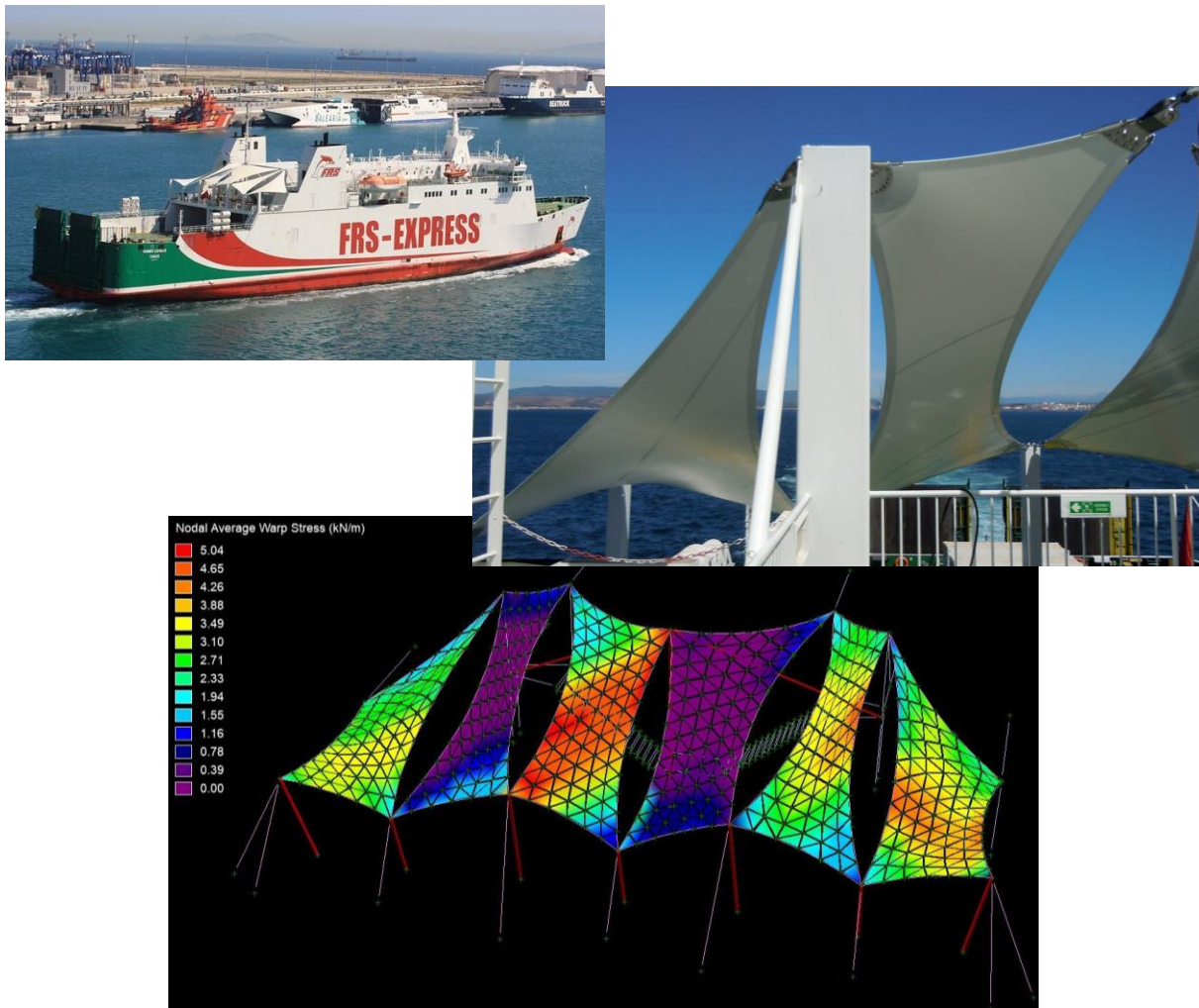

Figure 17 – Volvo Pure Tension Pavilion

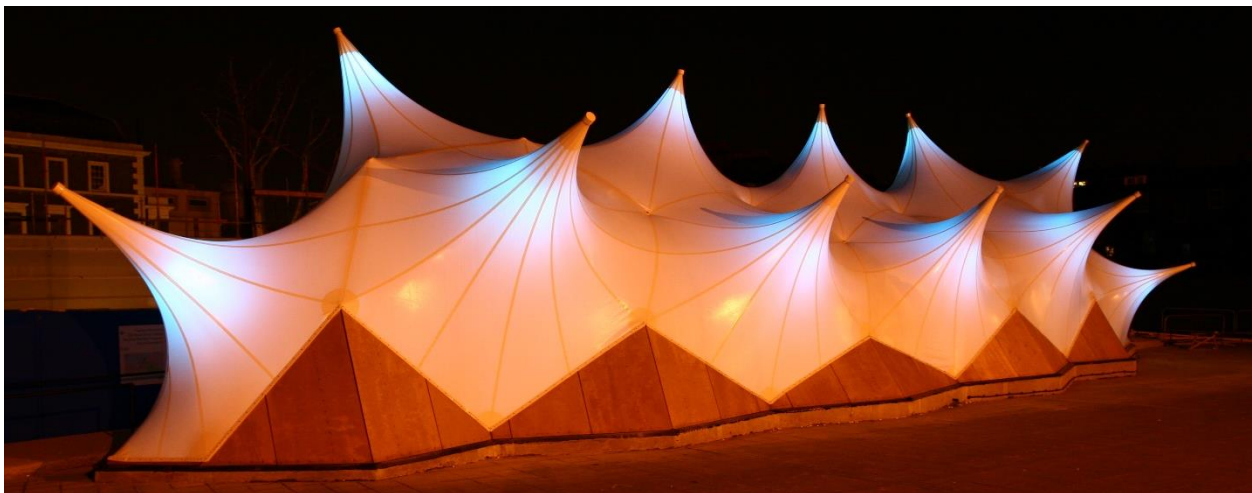Figure 18 – Passenger Deck Cover, Tanger Express, Spain

Figure 19 – Cutty Shark Clipper Ship, London, England

Figure 20 – Whitsunday Shopping Center, Queensland, Australia

Figure 21 – Cynthia Woods Concert Pavilion, Houston, Texas, United States.