

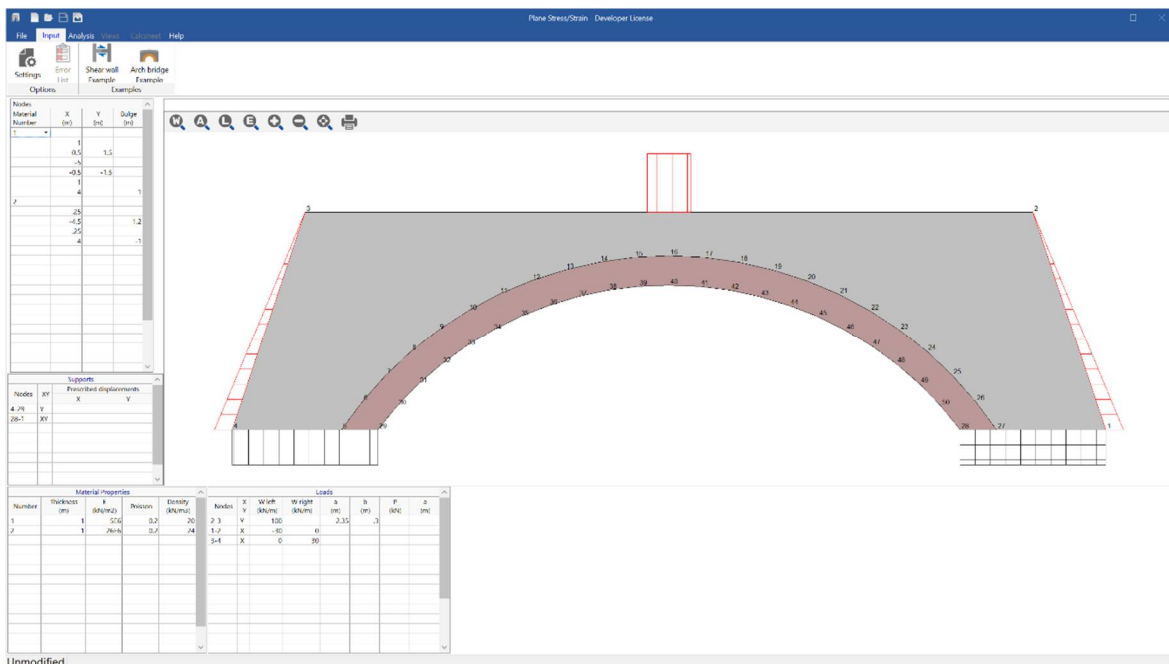


PLANE STRESS

FINITE ELEMENT ANALYSIS OF PLANE STRESS OR STRAIN PROBLEMS
ANALYSIS | A03

Summary

The **Plane Stress** module can be used to perform finite element analysis of a membrane of any general geometry subjected to plane stress or strain. You can combine simple polygons to model complex geometries, and the program automatically generates a suitable mesh for the analysis. Multiple load cases comprising point loads and UDL's can be applied to the outline of the model.



What makes this module special?

- Plane stress and plane strain problems can be solved
- Easy geometry input
- Automatic mesh generation

Detailed Description

Many complex three-dimensional problems are amenable to simplification to plane elasticity problems, e.g., regions of a dam wall far away from the edges can be analysed as a slice of unit thickness. Plane strain problems, such as the dam wall example, as well as plane stress problems, are the domain of **Plane Stress**.



PLANE STRESS

FINITE ELEMENT ANALYSIS OF PLANE STRESS OR STRAIN PROBLEMS
ANALYSIS | A03

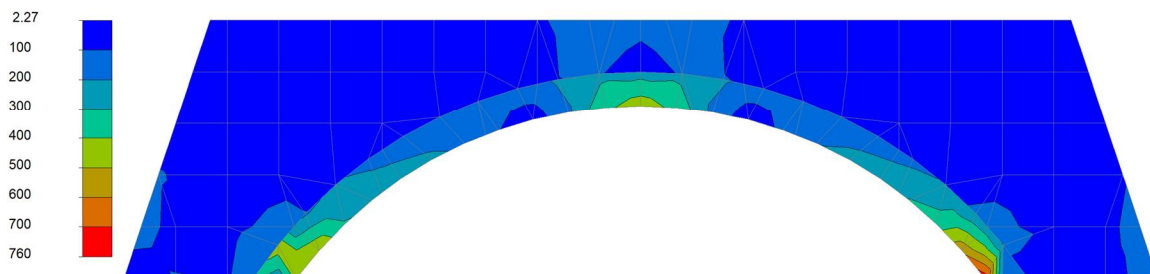
You can deal with both classes of problems in either **Sumo** or **Frame**, although plane strain requires modification of the elastic constants, the simplicity of Plane Stress makes it an attractive light-weight alternative.

The structure can be defined by entering one or more shapes comprised of straight lines and arcs. When more than one shape is entered the shape will accumulate and form one structure. Often, a complicated section is easily defined using more than one simple shape.

Point supports, distributed supports and prescribed displacements can be defined anywhere along the edges of the structure, as well as point loads and distributed loads. As many lines as necessary can be used to define the loads.

During the analysis, the program generates a rectangular grid of nodes in which rectangular and, where necessary, triangular finite elements are placed. The grid spacing can be set independently in the horizontal and vertical directions.

Smax= 760kPa @ x=0.000m, y=0.000m Smin= 2.27kPa @ x=0.400m, y=1.500m



The following analysis results can be viewed and printed in tabular or graphic format under the View tab:

- Maximum elastic deflections
- Maximum and minimum stress
- Von Mises stresses
- Stress vectors

Key features

- Plane strain and plane stress
- Simple input
- Automatic mesh generation