

2D Frame Analysis Program

Created by Antony Dewar as part of a dissertation project for civil & structural engineering at the University of Manchester, under supervision of Dr Parthasarathi Mandal

This page: Information & Features
The other pages: Screenshots/Example

Information

The program is capable of elastic analysis of user defined frame structures, with a visual representation of their structure. The program was written using C# and Visual Studio 2010, so won't work on an apple machine, but works on windows machines if the .NET Framework 4 is installed ([download link](#)). The program utilises the direct stiffness method and solves sets of simultaneous equations for the displacement of nodes, which are used to calculate member forces.

Features

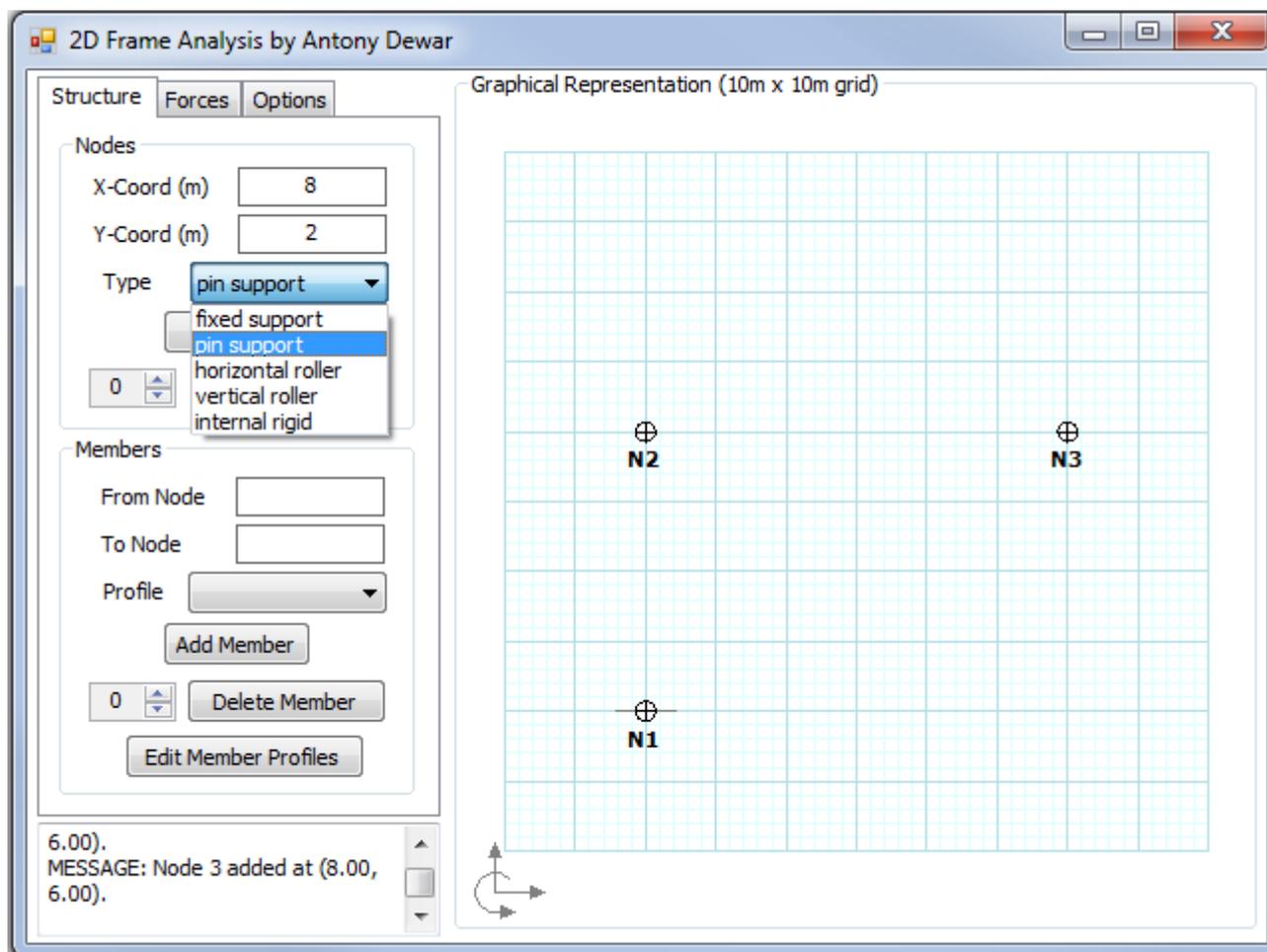
There may or may not be future updates, but currently using the program you can:

- Choose a grid size, reflecting the size of the structure (from 10m x 10m to 100m x 100m)
- Add nodes by their coordinates, (0,0) is the bottom left of the grid. There are 5 node types: fixed and pinned supports; vertical and horizontal rollers, and internal rigid connections. There isn't the option to add internal pins.
- Add members by selecting what node you want it to start and finish at and a member profile, member profiles can be created and edited for custom values of Young's Modulus of Elasticity, 2nd Moment of Area and Cross Sectional Area.
- Add forces to the structure. Point loads in the x and y directions can be applied directly to nodes, as well as bending moments. Uniformly distributed loads can also be added directly to members.
- Delete members and nodes, or the entire structure. There are a variety of visibility options too. You can also resize the window to make the drawn image larger.
- Calculate reaction forces at restricted nodes, as well as displacements in free degrees of freedom.
- View the shear force, axial force and bending moment diagrams.
- Get all the calculated data printed to a notepad file, including:
 - Location and type of all nodes, any rotation/displacements and any applied forces in any degrees of freedom, as well as forces as a result of member loading.
 - A list of all the member profiles and their properties.
 - A list of all members, which nodes they span from and to, their length, the angle they create with the global x-axis, and their end shear and axial forces and bending moments.
 - The **global stiffness matrix**, and all of the local member stiffness matrices.
- A couple of limitations: the program has a maximum node count of 50, and there aren't currently any saving features. There are probably more.

Screenshots

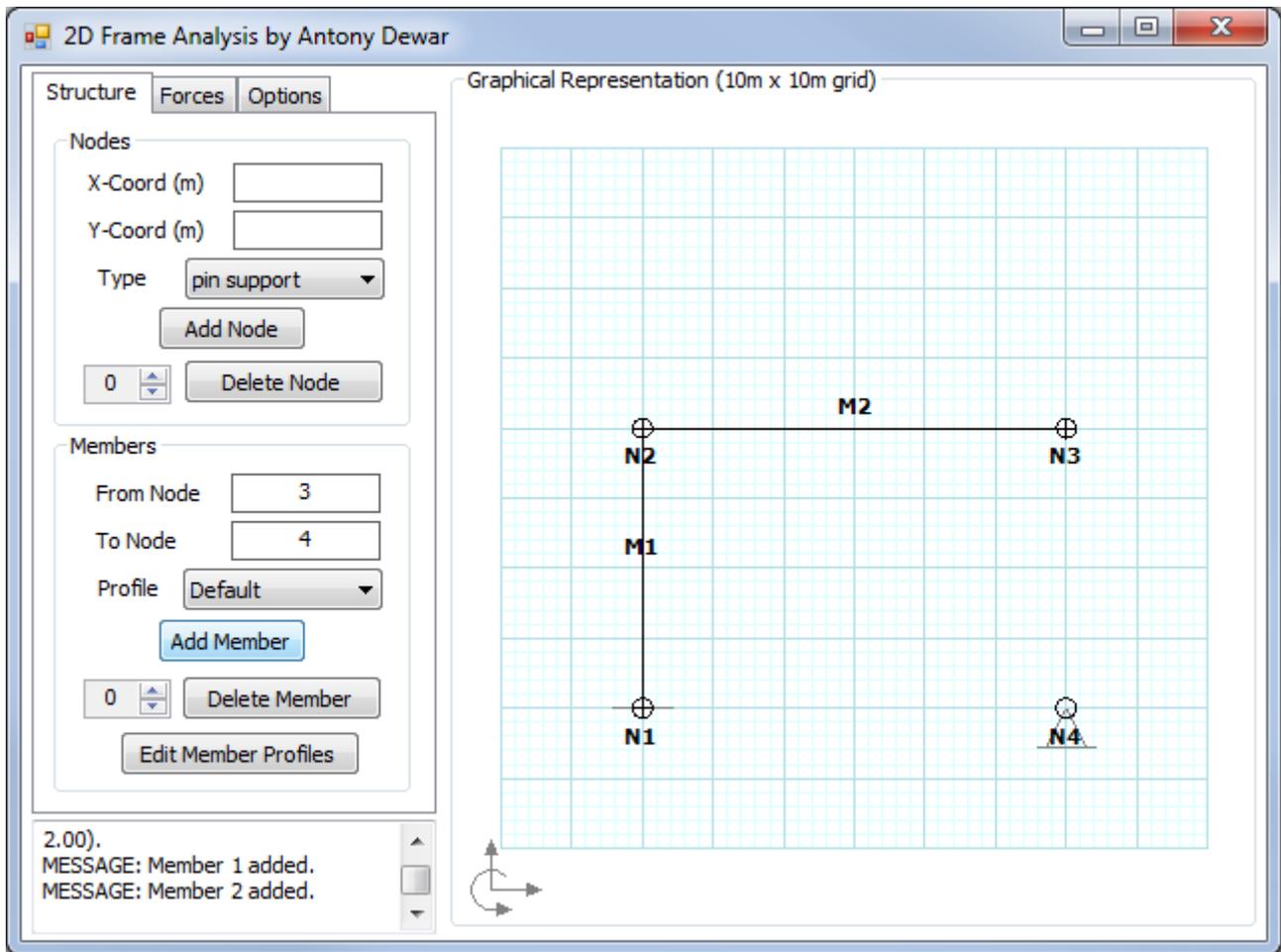
The following screenshots show how to analyse a portal frame using the program.

After choosing a grid size, (in this case 10m x 10m) the first step is to add nodes.

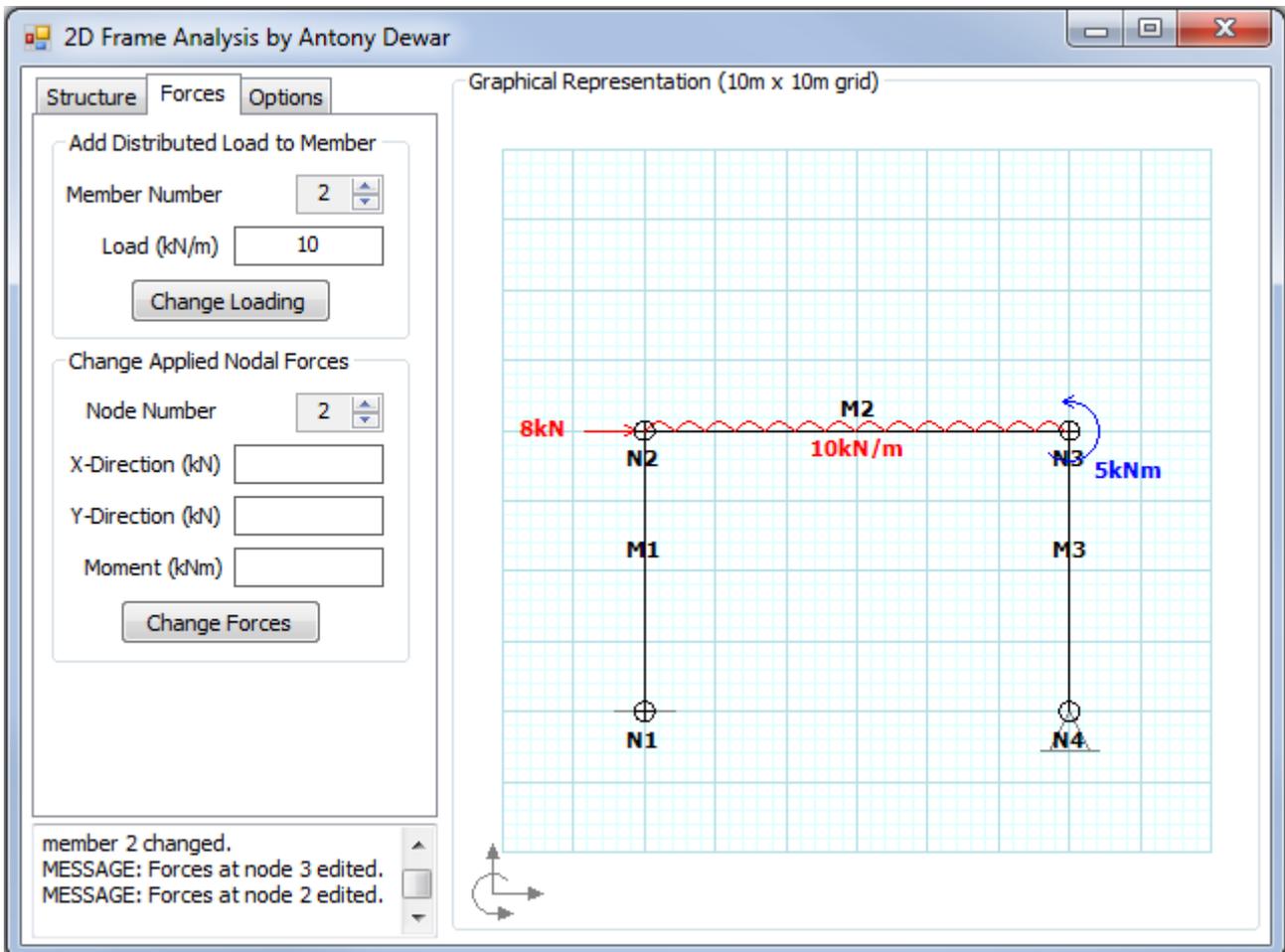


A fixed support has been added at (2, 2) and two internal rigid nodes have been added at (2, 6) and (8, 6). A final pin support is being added at (8, 2).

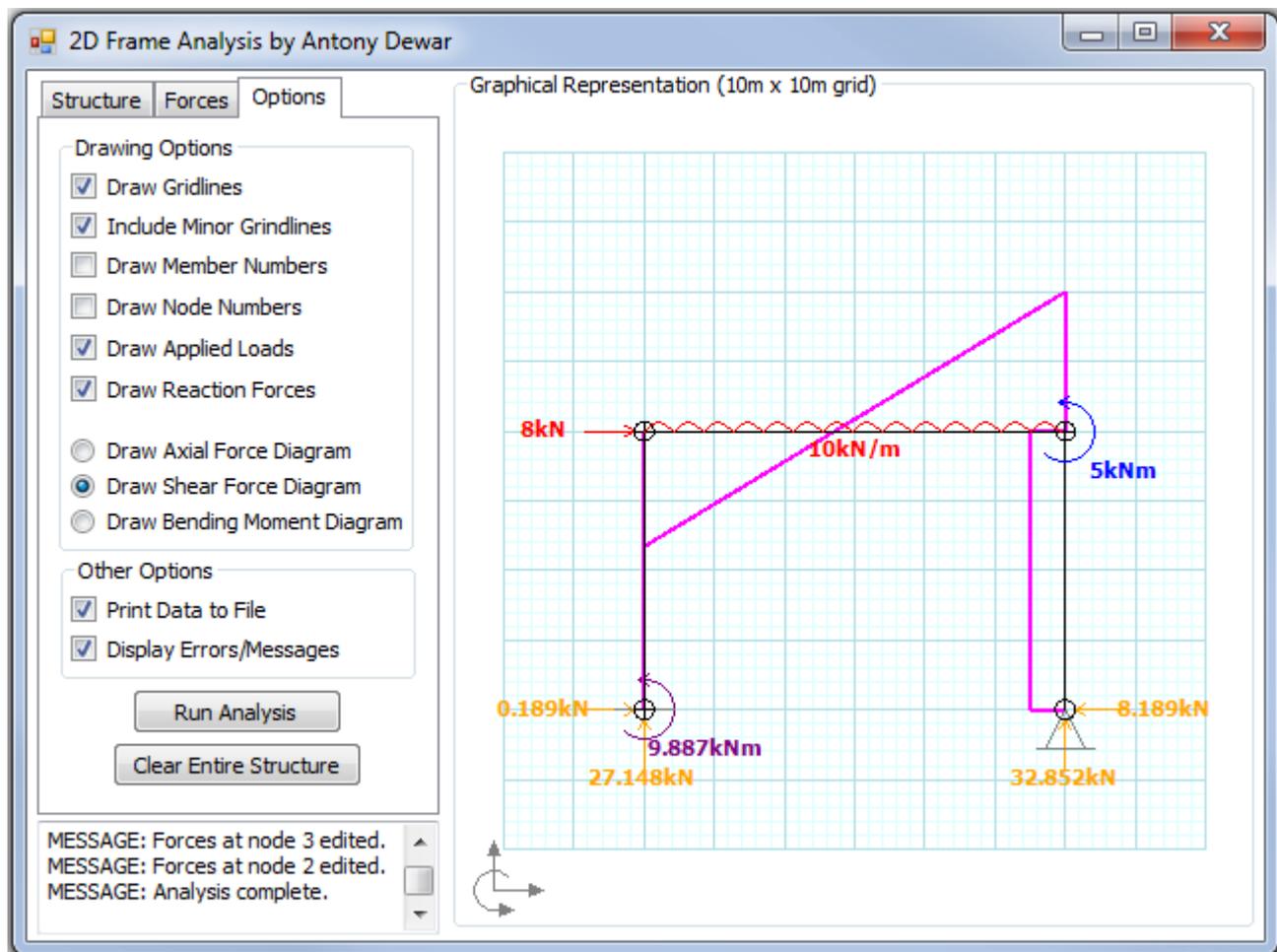
A member can be drawn between any two nodes; members can only be connected at nodes. Custom member profiles can be defined in a separate window, and then the custom member profiles can be selected from the drop down box. Members and nodes can be deleted by selecting the member or node number and clicking "delete member/node".



Once the members have been added, member loads can be applied to them.



Point loads are displayed in red and moments are displayed in blue. On the forces tab the forces at each node can be changed. Forces for a degree of freedom will only be changed if the box isn't left blank, to delete a force in the relative degree of freedom, just type "0" and then "change forces". To add point loads to the centre of members, create two half-members instead (sorry.) The sign convention for distributed loads is the opposite of what it normally is, so the applied load in the third screenshot is pointing downwards (negative in the x-direction) but was just input as "10" rather than "-10".



When you click "Run Analysis" on the options tab you can see the reaction forces and choose to look at the shear force, axial force and bending moment diagrams. All of the calculated data goes to a .txt file called "output" which saves to wherever the .exe is, as shown in the screenshot on the next page.

The program is free to [download](#) from my website. I hope you find it useful, I love it and I hope you do too.

```
output - Notepad
File Edit Format View Help
Equivalent joint bending moment as a result of distributed loading = 3.0E01kNm
# Node 4 properties:
Type = pin support
X-Coordinate = 8m
Y-Coordinate = 2m
Rotation about z-axis = -8.355E-04rad
Reaction force in x-direction = -8.1891E00kN
Reaction force in y-direction = 3.2852E01kN
##### 02 Profiles #####
# Profile 1 "default" properties:
E = 2.0E02GPa
A = 9.0E03mm2
I = 2.5E08mm4
##### 03 Members #####
# Member 1 properties:
Spans from node 1 (i) to node 2 (j)
Profile = 1, "default"
Length = 4.0m
Angle = 1.570796rad
Axial force at end i = -2.7148E01kN, at end j = -2.7148E01kN
Shear force at end i = -1.8912E-01kN, at end j = -1.8912E-01kN
Bending Moment at end i = -9.8872E00kN, at end j = -1.0644E01kN
Local Member Stiffness Matrix:
+4.5000E+02      0      0      -4.5000E+02      0      0
  0      +9.3750E+00 +1.8750E+04      0      -9.3750E+00 +1.8750E+04
  0      +1.8750E+04 +5.0000E+07      0      -1.8750E+04 +2.5000E+07
-4.5000E+02      0      0      +4.5000E+02      0      0
  0      -9.3750E+00 -1.8750E+04      0      +9.3750E+00 -1.8750E+04
  0      +1.8750E+04 +2.5000E+07      0      -1.8750E+04 +5.0000E+07
Transformed Member Stiffness Matrix:
+9.3750E+00      0      -1.8750E+04 -9.3750E+00      0      -1.8750E+04
  0      +4.5000E+02      0      0      -4.5000E+02      0
-1.8750E+04      0      +5.0000E+07 +1.8750E+04      0      +2.5000E+07
-9.3750E+00      0      +1.8750E+04 +9.3750E+00      0      +1.8750E+04
  0      -4.5000E+02      0      0      +4.5000E+02      0
-1.8750E+04      0      +2.5000E+07 +1.8750E+04      0      +5.0000E+07
# Member 2 properties:
Spans from node 2 (i) to node 3 (j)
Profile = 1, "default"
```