

Basic Outline of Setting Up an Ice Sheet in ADINA-F - in general terms and with a specific example. Tony Gades, UW Geophysics Spring 2001

Download this document as an Adobe Acrobat .pdf file:

The best way to learn some of the idiosyncrasies of ADINA is to start right in. These instructions are specific to ADINA V7.4. Earlier versions work differently and probably later versions will change some things as well. The good thing is that the changes I have seen in the past year of working with this program have been for the better.

General Overview:

ADINA - (Automatic Dynamic Incremental Nonlinear Analysis) has several parts to the overall package. The relevant portions of the package to ice flow modeling are:

ADINA-F: for FEM models of fluids

ADINA-T: for doing heat flow

ADINA-PLOT: for viewing the results of a model.

I start by building a basic ice sheet in ADINA-F. This model will be a basic ice sheet based on actual Roosevelt Island bed/surface topography. The first goal will be to set up an ice sheet and just let it flow - without accumulation. Then we add accumulation and finally heat flow.

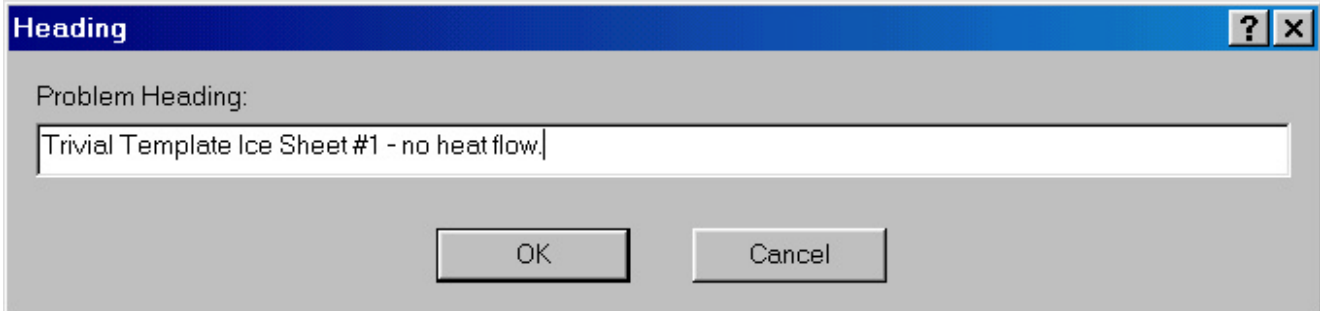
A Few General Tips :

- 1) As with any program, save often.
- 2) The tiny icons in the ADINA gui are not that useful in my opinion but are redundant with menu options. I opt for the menu options in describing steps.
- 3) There are many ways to do many of the things I describe here. I have tried many different ways of doing things and these steps are just what I estimate to be the best or simplest.
- 4) ADINA allows you to create a simple ASCII text script file of the commands that produce your model (a ".in" file). It seems as though one can get only so far by using the menu options/icons in the gui and at some stage one must directly edit the .in file. For example, I could not determine any other way to specify a velocity function aside from directly editing the command file. A combination of using the ADINA-gui and editing the resulting command file seems to be most efficient.
- 5) If you find it simpler to just have a look at a command script file to figure out how this model works, here is one. [RI IceSheet NoAccumulation.in](#)

HERE ARE THE STEPS:

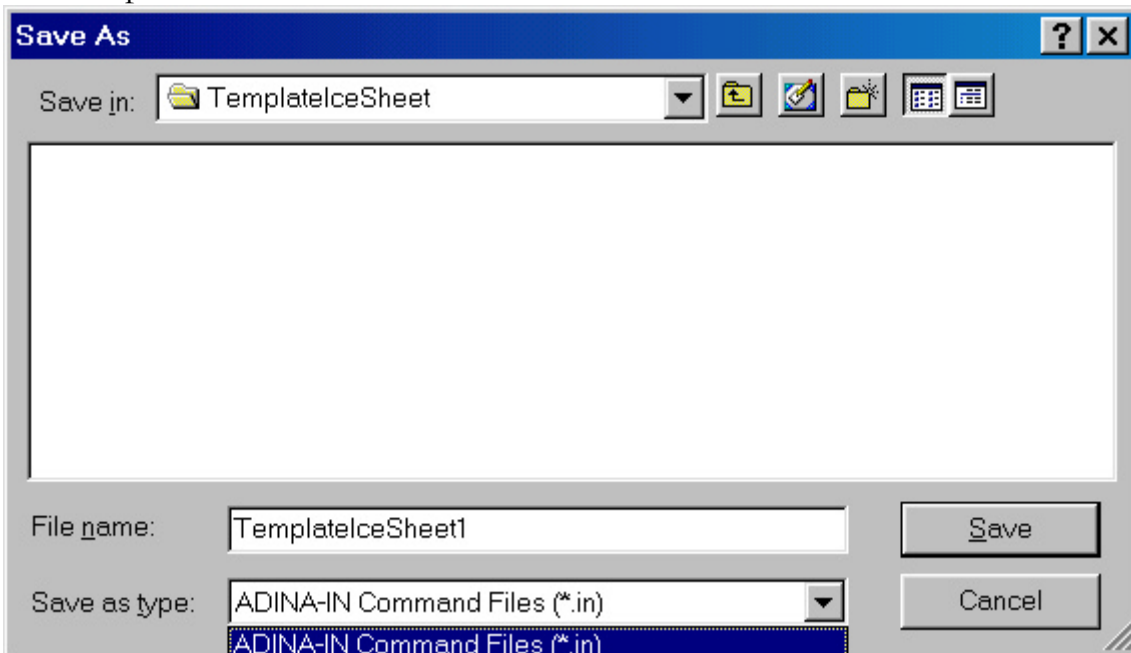
1) Give the model a descriptive title and Save the File:

The heading is just a label - sort of like a comment at the beginning of some code. In ADINA-F, define a heading by Menu Option: **Control->Heading** (this means click on the Control menu, then select the Heading option)



Save the file as a ".in" file. This yields an ASCII text file of all of the commands and allows us to edit the thing later - this is not the default.

Menu Option: **File->SaveAs**



Choose "Save As Type" ADINA-IN Command Files.

2) Decide on a geometry.

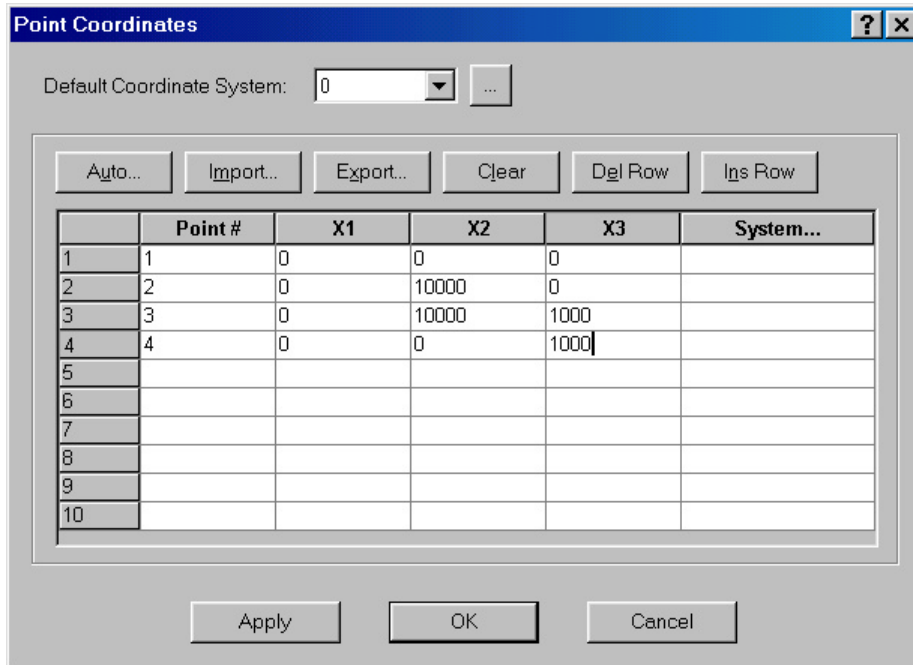
a) If complex geometry, then use the template matlab file to produce a file in ADINA.in format.

b) If simple geometry, then just make a sketch and make a table of xy or yz points. It is best to label the points clockwise or counterclockwise around the geometry of the ice sheet.

Here is a very simple example in the YZ plane with Z positive upward, Y positive to the right. 1000 meters thick, flat surface, 10000 meters wide.

Point#	X	Y	Z
1	0	0	0
2	0	10000	0
3	0	10000	1000
4	0	0	1000

Here is how you actually define points from within ADINA-F. Menu option: **Geometry->Points**



3) Define the lines that outline the geometry of the ice sheet.

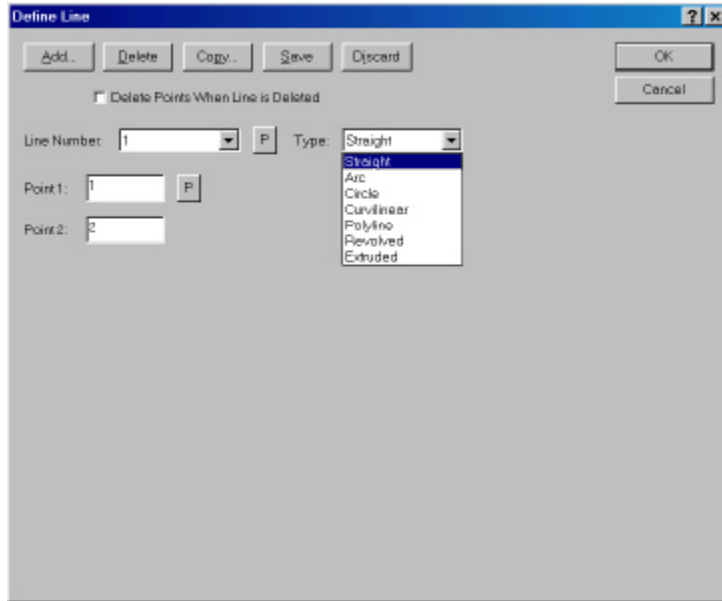
Complex geometries require using more complex line types. For example, the "polyline" type allows you to connect (via linear or other types of interpolation) many points. This is what I use for defining actual ice surfaces and bed.

Very simple ice sheets like the one above has 2 cliff edges, a surface and a bed requires defining

4 simple lines that are defined only by the 2 end points.

Line#	Point1	Point2	
1	1	2	(bed)
2	2	3	(RHS cliff)
3	3	4	(Surface)
4	4	1	(LHS cliff)

In ADINA-F, you define lines from: **Geometry->Lines->Define**
Click "ADD" button and then fill in the information:

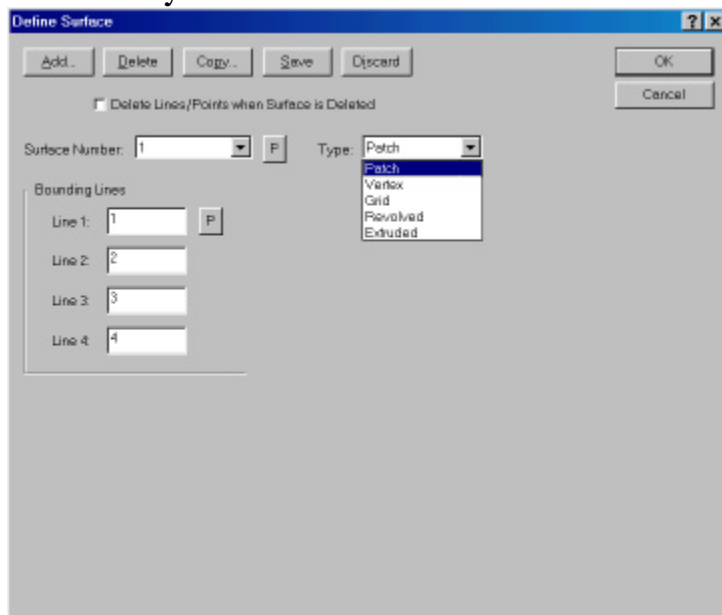


Instead of typing in the points, you can click the "P" box to the right of the "point 1" entry, this will take you to the main ADINA screen and then you can just click on the two points that you wish to use to define line 1. Click "OK" when done and then "ADD" again and repeat for all 4 lines. There are many line types but the 2 I find most useful are "straight" and "polyline".

4) Define the 2 -D surface or surfaces that define the model.

The "patch" type surface works well for defining the 2-d surface. Carrying on with the simple ice sheet, one can define a patch that is bounded by the 4 lines. The "surface" in the 2-D case refers to the body of the ice sheet.

In ADINA-F **Geometry->Surfaces->Define**



Again, there are many ways to define a surface but a patch of 4 lines works well.

5) Define the Material Model properties

To read the details of how I got help sorting this out, click on this link: materialmodel.html

Otherwise, you can just try this for -10C ice:

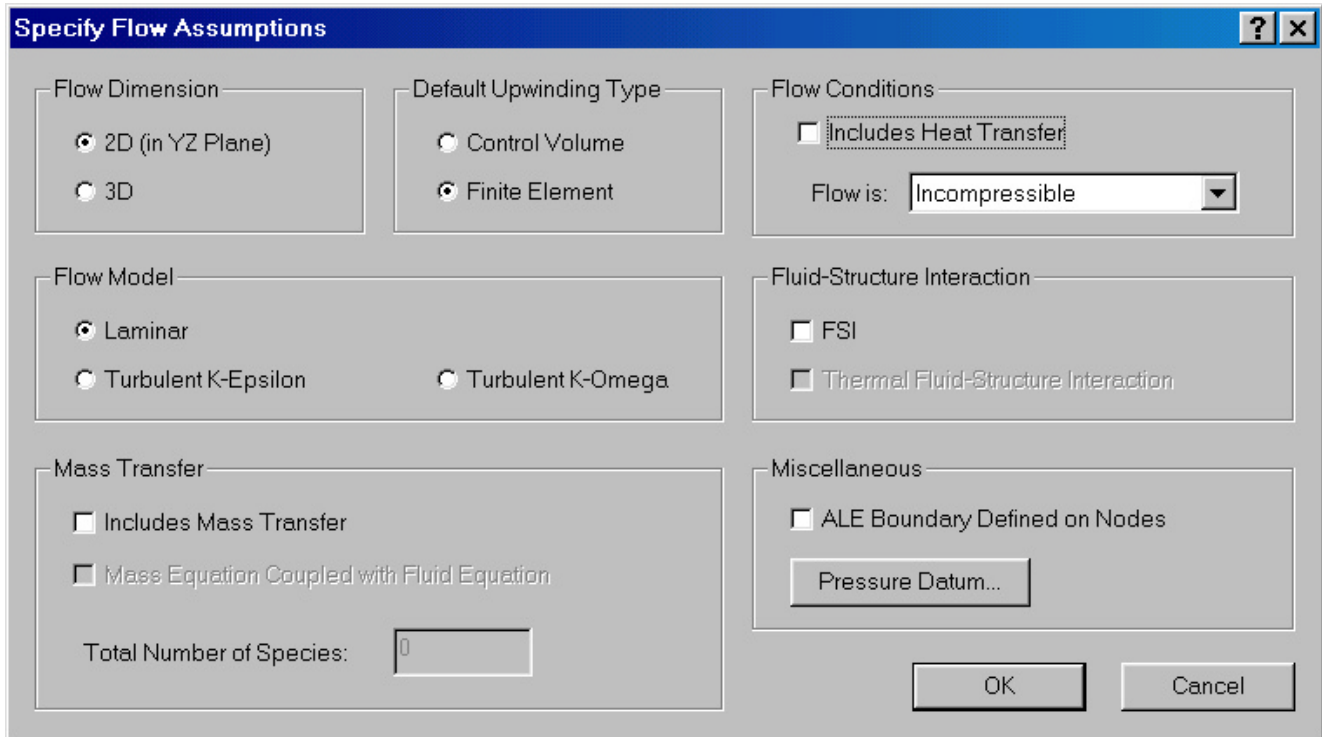
Menu option: **MODEL->MATERIALS->POWER LAW->ADD**

The screenshot shows the 'Define Power-Law Material' dialog box. The title bar is blue with a question mark and a close button. Below the title bar are buttons for 'Add...', 'Delete', 'Copy...', 'Save', 'Discard', 'OK', and 'Cancel'. The 'Material Number' is set to 1. The 'Constant, Mu 0' is 1.0204e16, 'Density' is 917, 'Constant, A' is 6.3422e7, 'Heat Capacity' is 0, 'Power Law Exponent' is -0.6666667, 'Thermal Conductivity' is 0, 'Coef. of Volume Expansion' is 0, 'Rate of Heat Generated Per Unit Volume' is 0, 'Reference Temperature' is -10, 'Coef. of Surface Tension' is 0, and 'Fluid Bulk Modulus' is 1e+020. The 'Acceleration due to Gravity' section has 'x: 0', 'y: 0', and 'z: -9.8'.

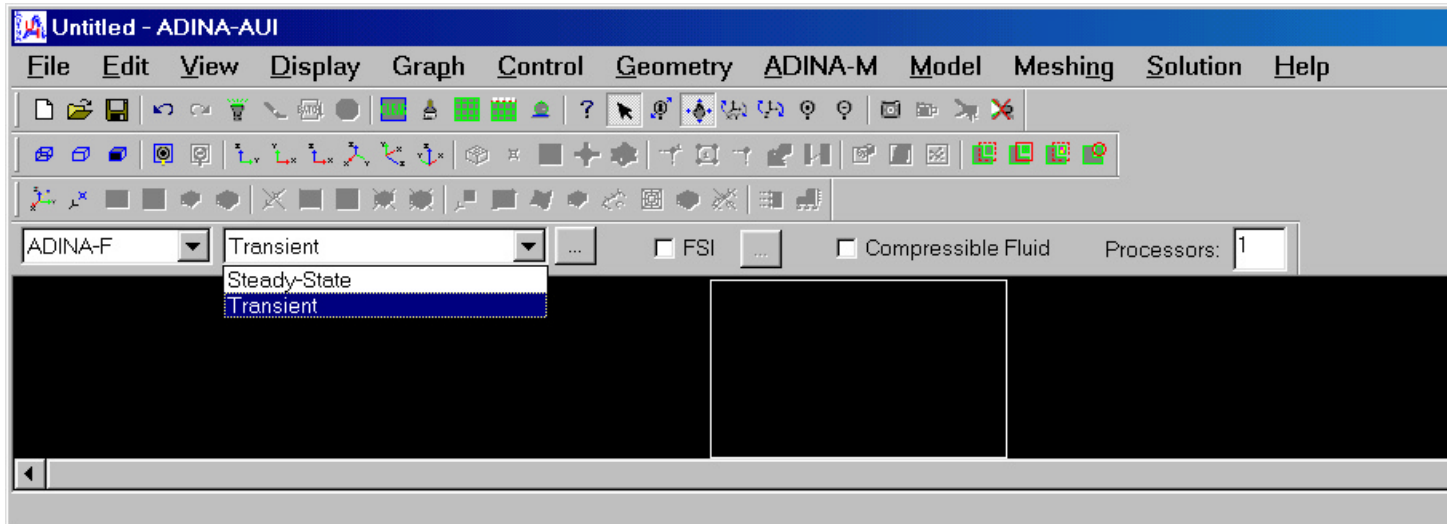
Or you can just try this for -25C ice:

The screenshot shows the 'Define Power-Law Material' dialog box. The title bar is blue with a question mark and a close button. Below the title bar are buttons for 'Add...', 'Delete', 'Copy...', 'Save', 'Discard', 'OK', and 'Cancel'. The 'Material Number' is set to 1. The 'Constant, Mu 0' is 5.319e16, 'Density' is 917, 'Constant, A' is 1.1e8, 'Heat Capacity' is 0, 'Power Law Exponent' is -0.6666667, 'Thermal Conductivity' is 0, 'Coef. of Volume Expansion' is 0, 'Rate of Heat Generated Per Unit Volume' is 0, 'Reference Temperature' is -25, 'Coef. of Surface Tension' is 0, and 'Fluid Bulk Modulus' is 1e+020. The 'Acceleration due to Gravity' section has 'x: 0', 'y: 0', and 'z: -9.8'.

6) Define the flow assumptions; Set solution to transient
 Menu Option: **MODEL->FLOW ASSUMPTIONS**



Now click option to change to transient:



7) Define the mesh density

This is the step where you define how the mesh nodes are distributed within the model. There are many ways to do this, but this method allows you to specify that the nodes are preferentially stacked toward the divide and if desired toward the bottom. To do this I use the "Geometric" progression of element edge lengths.

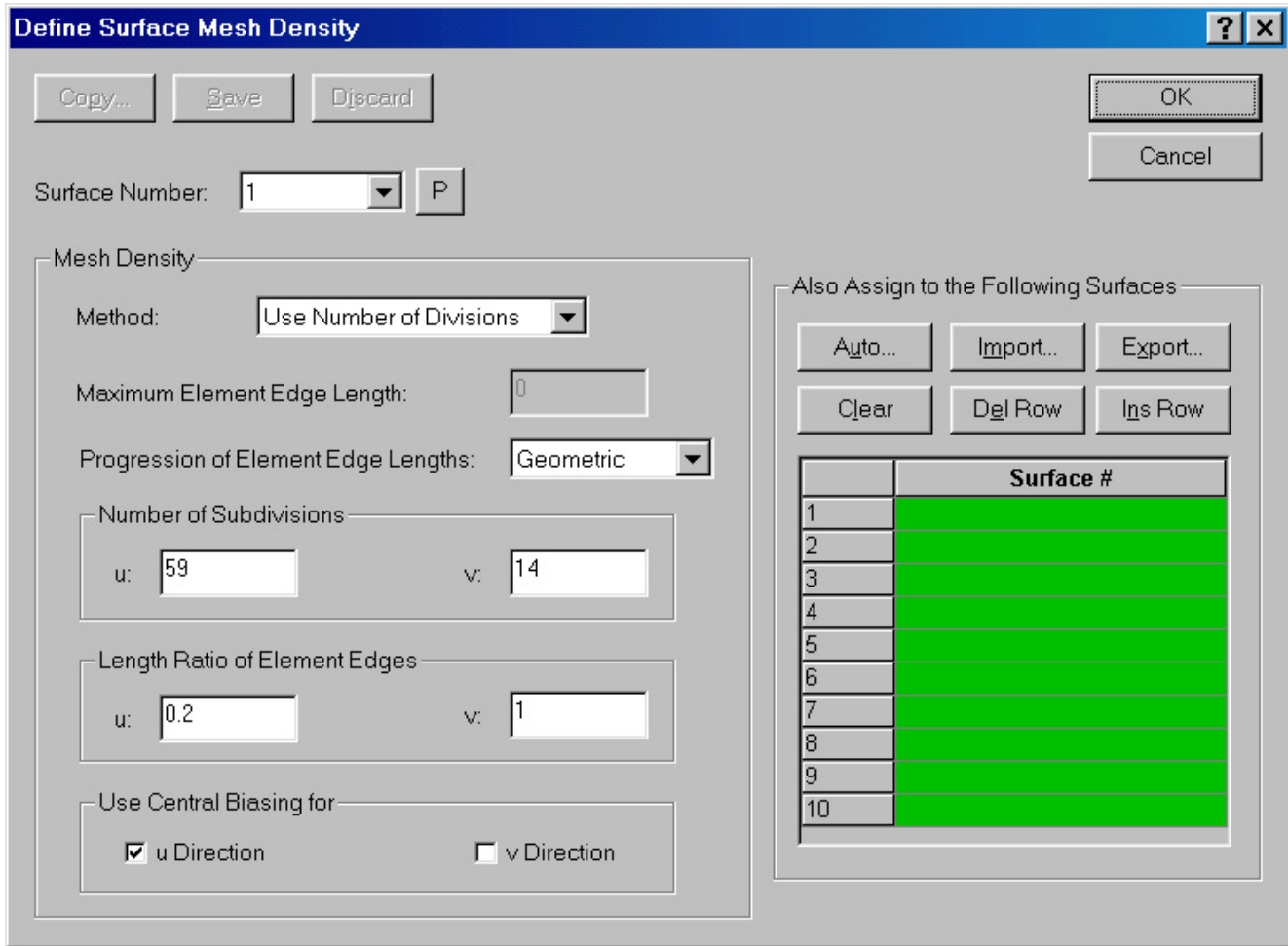
The example below divides the nodes evenly from top to bottom by:

- 1) Not selecting the central biasing in v direction.
- 2) Using a length ratio of 1 in the "v" direction.

The example preferentially places nodes toward the horizontal center by:

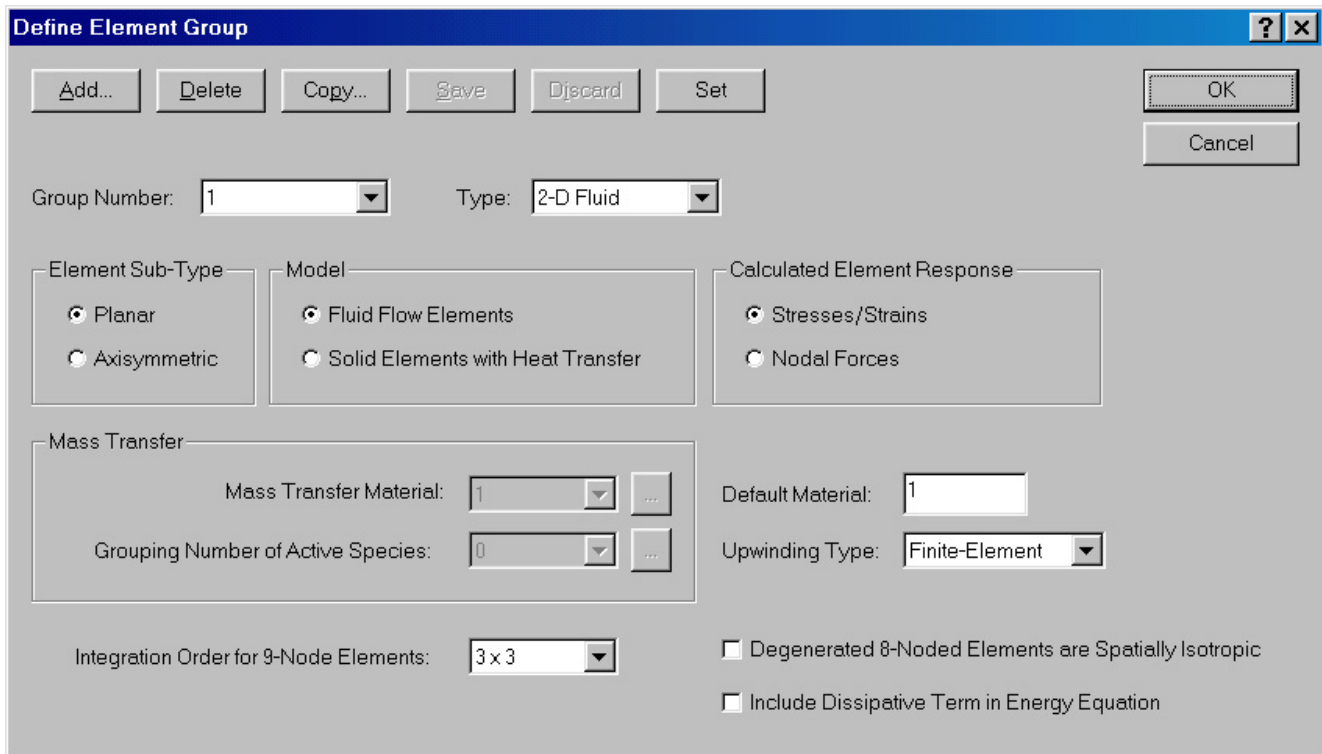
- 1) Use central biasing for the u Direction.
- 2) A value of less than 1 for the "Length ratio of element edges".

Menu Option: **MESHING->MESH DENSITY->SURFACE**



8) Define element type

Menu Option: **MESHING->ELEMENT GROUPS->ADD**

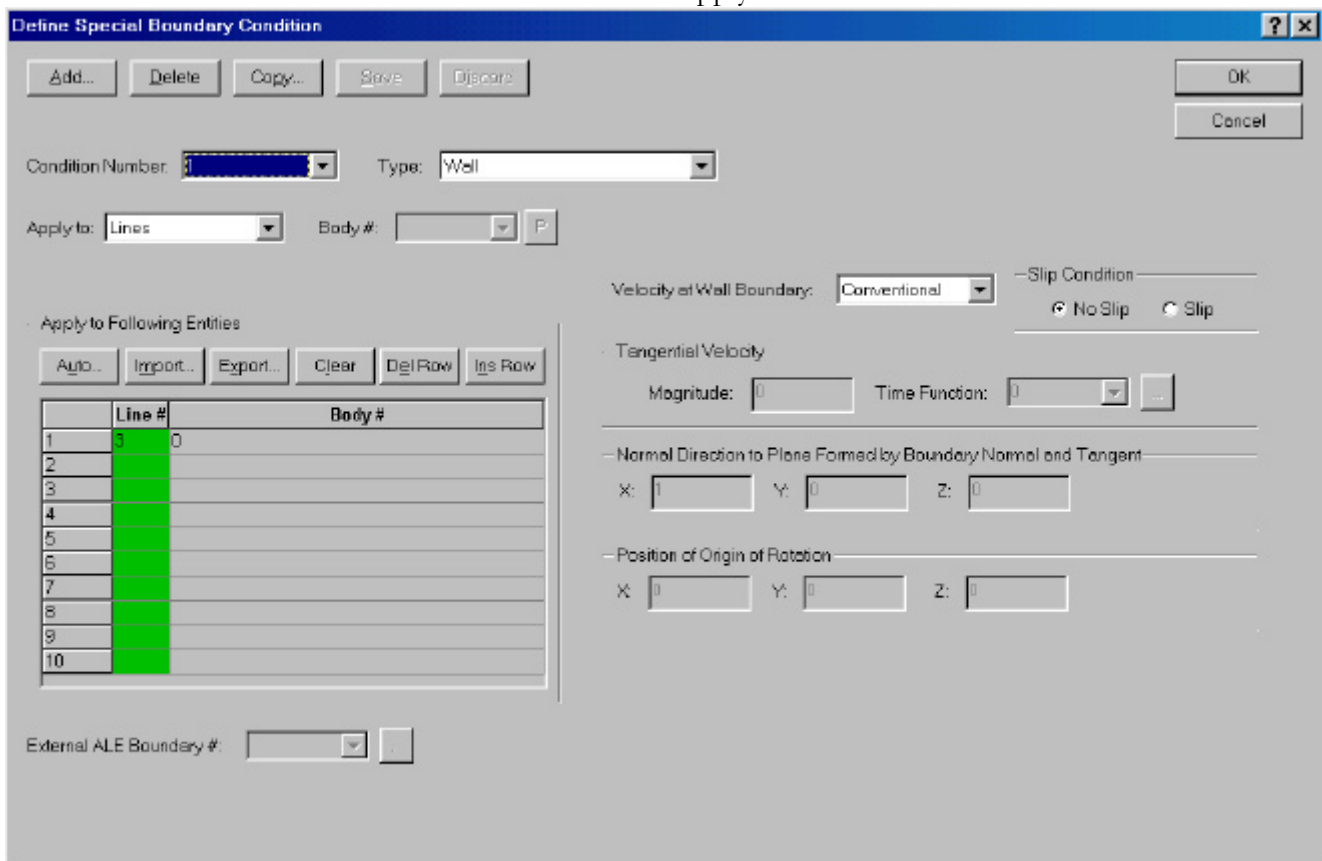


9) Define the boundary conditions for surface and bed

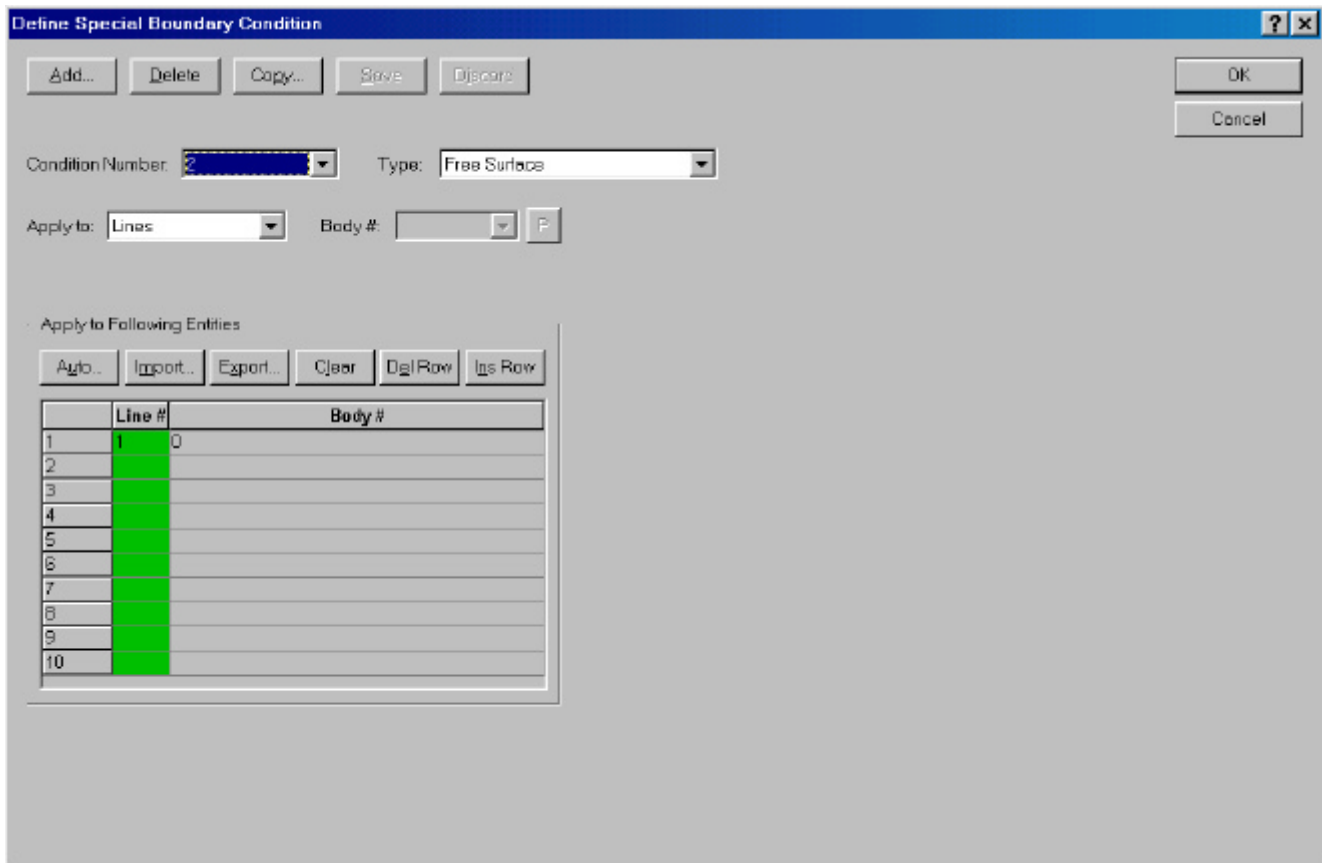
To begin with we want to have a free surface boundary condition for the ice sheet surface and a no-slip "wall" condition for the bed.

MENU OPTION: **MODEL->SPECIAL BOUNDARY CONDITIONS->ADD**

Add condition 1 for the wall condition of the bed and apply it to line 3.



Add condition 2 for the ice sheet surface and apply it to line 1:



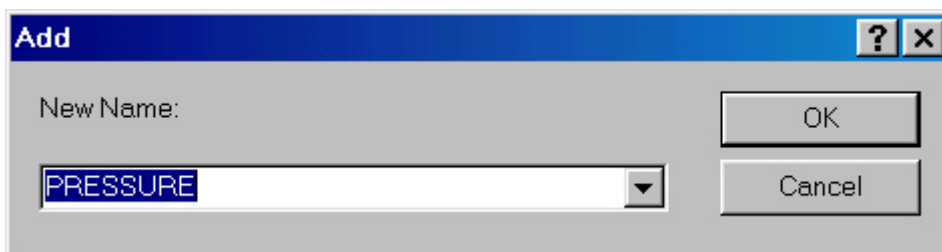
10) Pressure Fixity

For the model to arrive at a solution, you have to define a pressure zero - this is arbitrary and where you choose to set the zero doesn't seem to affect the model in any way :

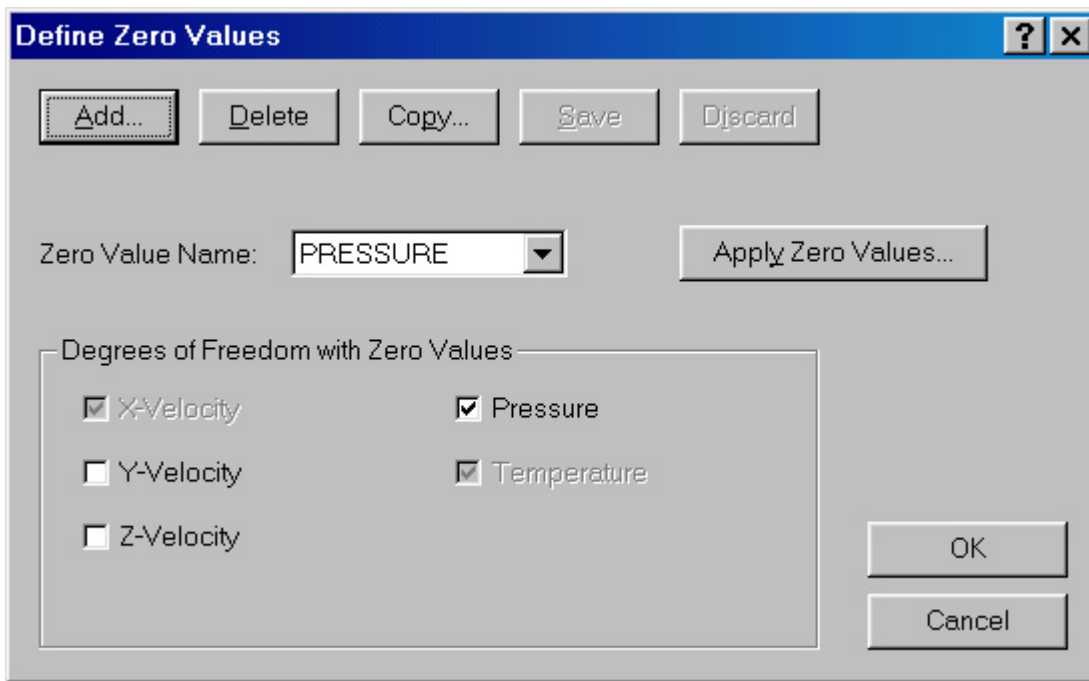
MENU OPTION:

MODEL->USUAL BOUNDARY CONDITIONS/LOAD->ZERO VALUES->ADD

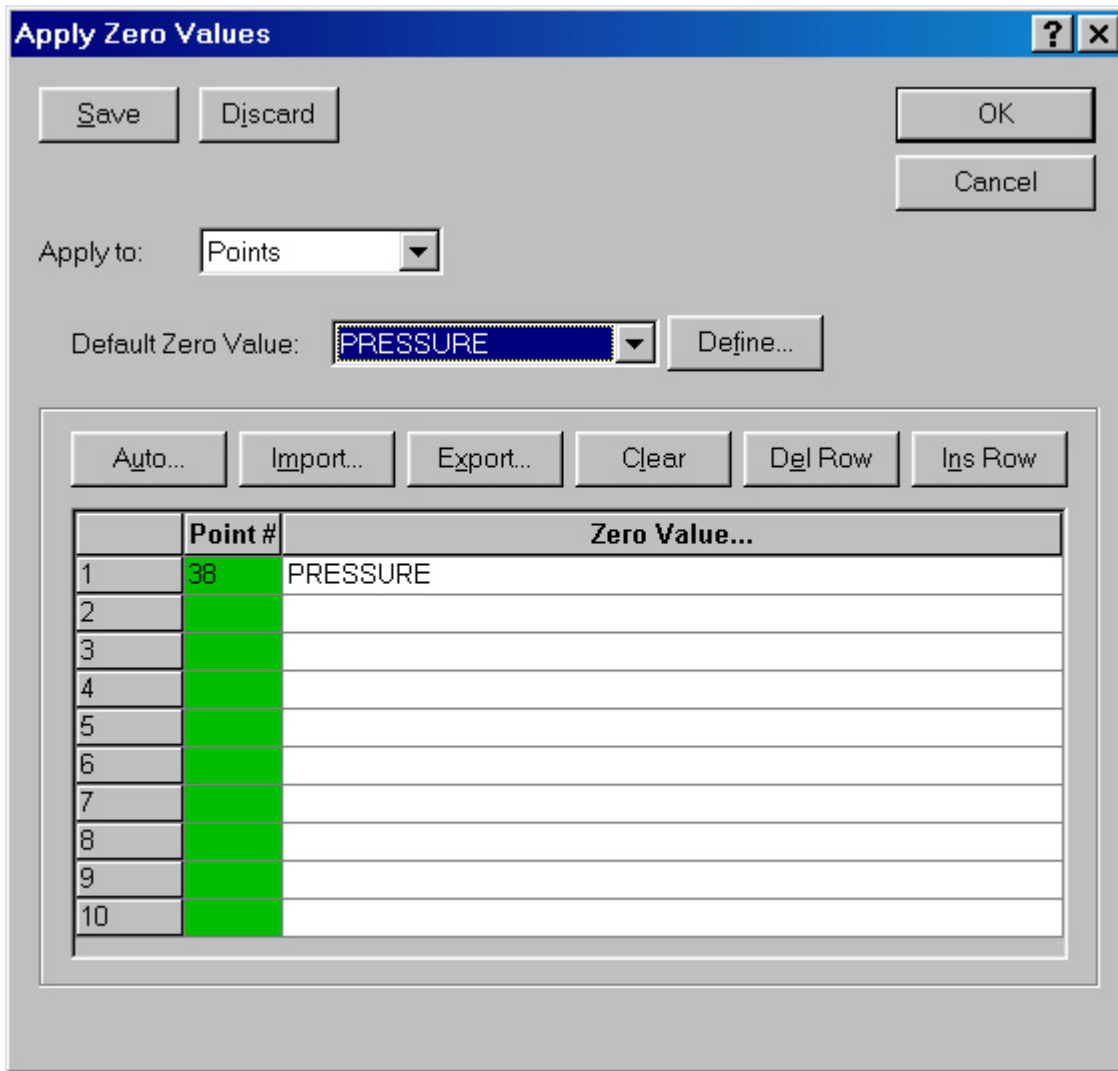
We need to define a new type of zero value so lets name it PRESSURE.



Then define the properties of this new zero:



Select only the "Pressure" box as above, Then click "**Apply Zero Values**" and apply the zero to some point on the model - either type in a point number or click in the green box with the mouse and it will allow you to select a point, hit escape to come back to this box. Make sure that the zero applied is the PRESSURE zero that we just defined as shown below.



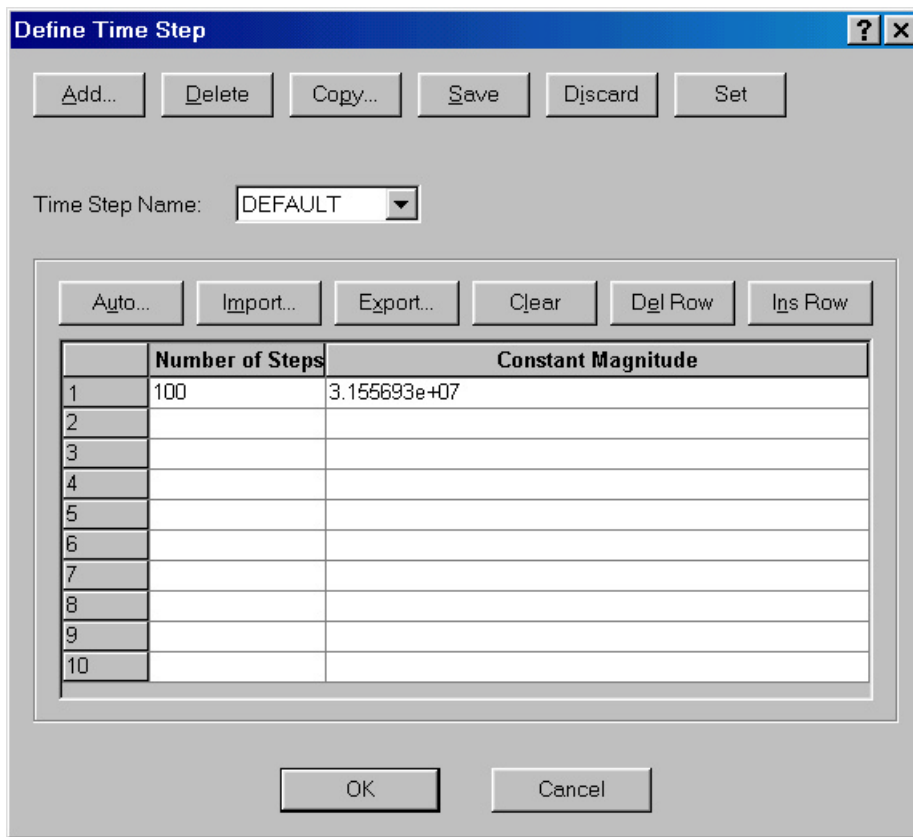
11) Define Time steps

I set this entire model up in MKS units. This means time is in seconds.

1 year = **3.155693e+07** seconds

So, that is the magnitude needed for a single year time step. To start, set up the model for 100 steps of 1year each.

MENU OPTION: CONTROL ->TIME STEP



12) Add equilibrium velocity conditions at margin. (INCOMPLETE)

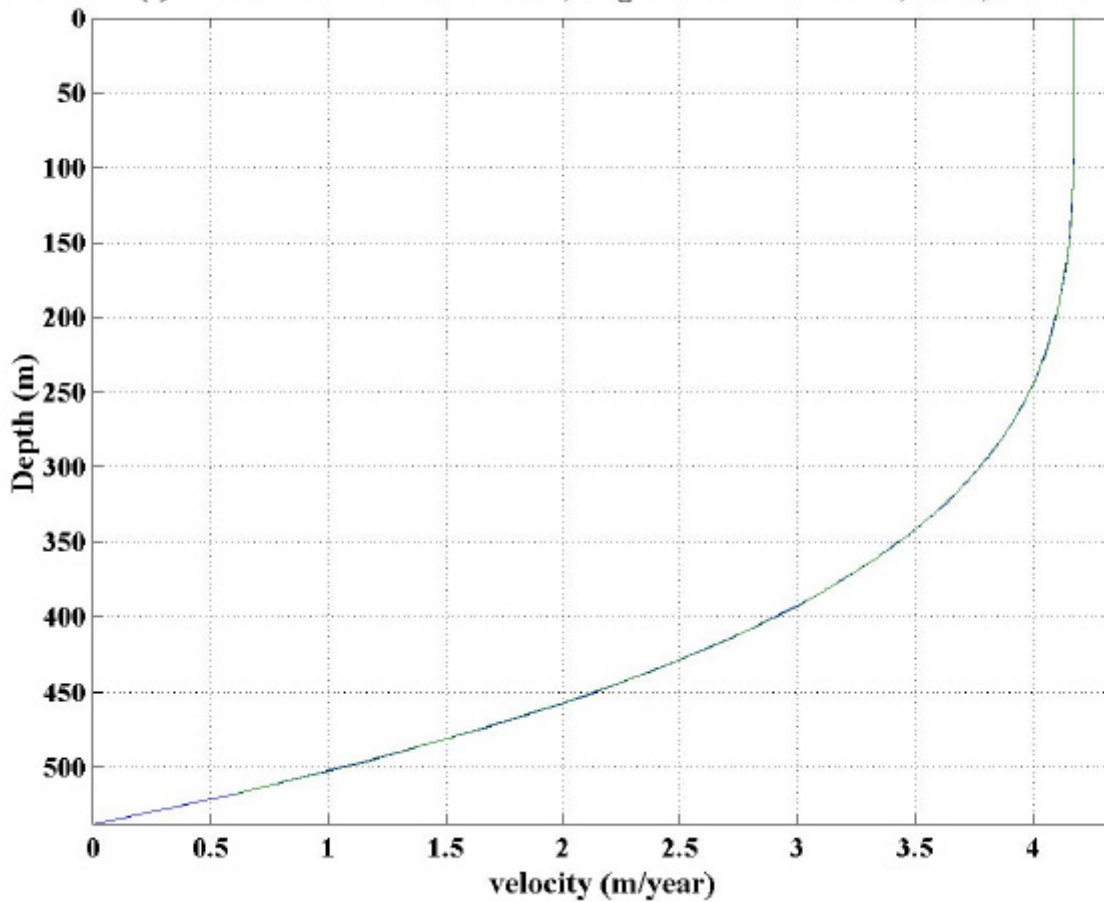
In this next and final step, we have to prescribe a velocity profile at the margins of the ice sheet that give equilibrium assuming some accumulation rate. That is, a velocity profile that gets rid of an amount of ice equal to sum total added at the surface. Paterson, Chapter 11, details all of this.

The logistics of adding this velocity profile are much easier to do if you actually edit the command file - I don't know of any better way to do this.

The general idea is that first you specify a "line-function" as a table of values and then you apply a "velocity load" to a line in your model. The table can include as many values as desired - the more values, the higher resolution the function you specify. This method is a bit clunky but appears to be the only option.

So, if I want to specify a velocity function like this for example:

Equilibrium $u(z)$ for Accumulation Rate=0.10, EdgeThicknesses=538.5,517.8, HalfWidth=18000



I would do it by specifying a table like this (in this case I arbitrarily have chosen a table length of 26 points):

Again, units are MKS so meters/second - an unfortunate set of units for ice flow...

```

*
* Ice Stream Edge Velocity Profile for equilibrium given
* Equilibrium  $u(z)$  for Accumulation Rate=0.10, EdgeThicknesses=538.5,517.8, HalfWidth=18000.0
*
LINE-FUNCTIO NAME=1 TYPE=TABULAR NPOINT=26
@CLEAR
1 1.324922e-007
2 1.324918e-007
3 1.324867e-007
4 1.324647e-007
5 1.324053e-007
6 1.322802e-007
7 1.320526e-007
8 1.316778e-007
9 1.311029e-007
10 1.302668e-007
11 1.291004e-007
12 1.275262e-007
13 1.254589e-007
14 1.228048e-007
15 1.194622e-007
16 1.153212e-007
17 1.102637e-007
18 1.041635e-007
19 9.688641e-008
20 8.828990e-008
21 7.822338e-008
22 6.652811e-008
23 5.303723e-008

```

```

24 3.757571e-008
25 1.996040e-008
26 1.126558e-022
@
*
LOAD VELOCITY NAME=1 VX=FREE VY=-1.00000000000000 VZ=FREE
*
* Velocity Profile for equilibrium given
* Equilibrium u(z) for Accumulation Rate=0.10, EdgeThicknesses=538.5,517.8, HalfWidth=18000.0
*
LINE-FUNCTIO NAME=2 TYPE=TABULAR NPOINT=26
@CLEAR
1 1.922721e-008
2 3.629097e-008
3 5.134995e-008
4 6.456770e-008
5 7.610079e-008
6 8.609884e-008
7 9.470451e-008
8 1.020535e-007
9 1.082746e-007
10 1.134895e-007
11 1.178130e-007
12 1.213531e-007
13 1.242107e-007
14 1.264796e-007
15 1.282469e-007
16 1.295926e-007
17 1.305898e-007
18 1.313045e-007
19 1.317960e-007
20 1.321164e-007
21 1.323109e-007
22 1.324179e-007
23 1.324687e-007
24 1.324875e-007
25 1.324919e-007
26 1.324922e-007
@
*
LOAD VELOCITY NAME=2 VX=FREE VY=1.00000000000000 VZ=FREE
*

```

Here is an example where these velocities are actually applied.

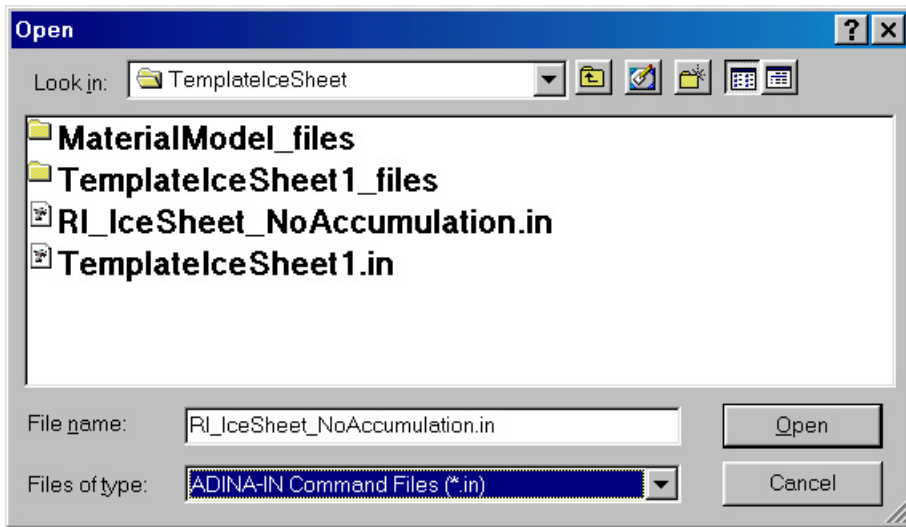
[RI IceSheet NoAccumulation.in](#)

13) TRY IT

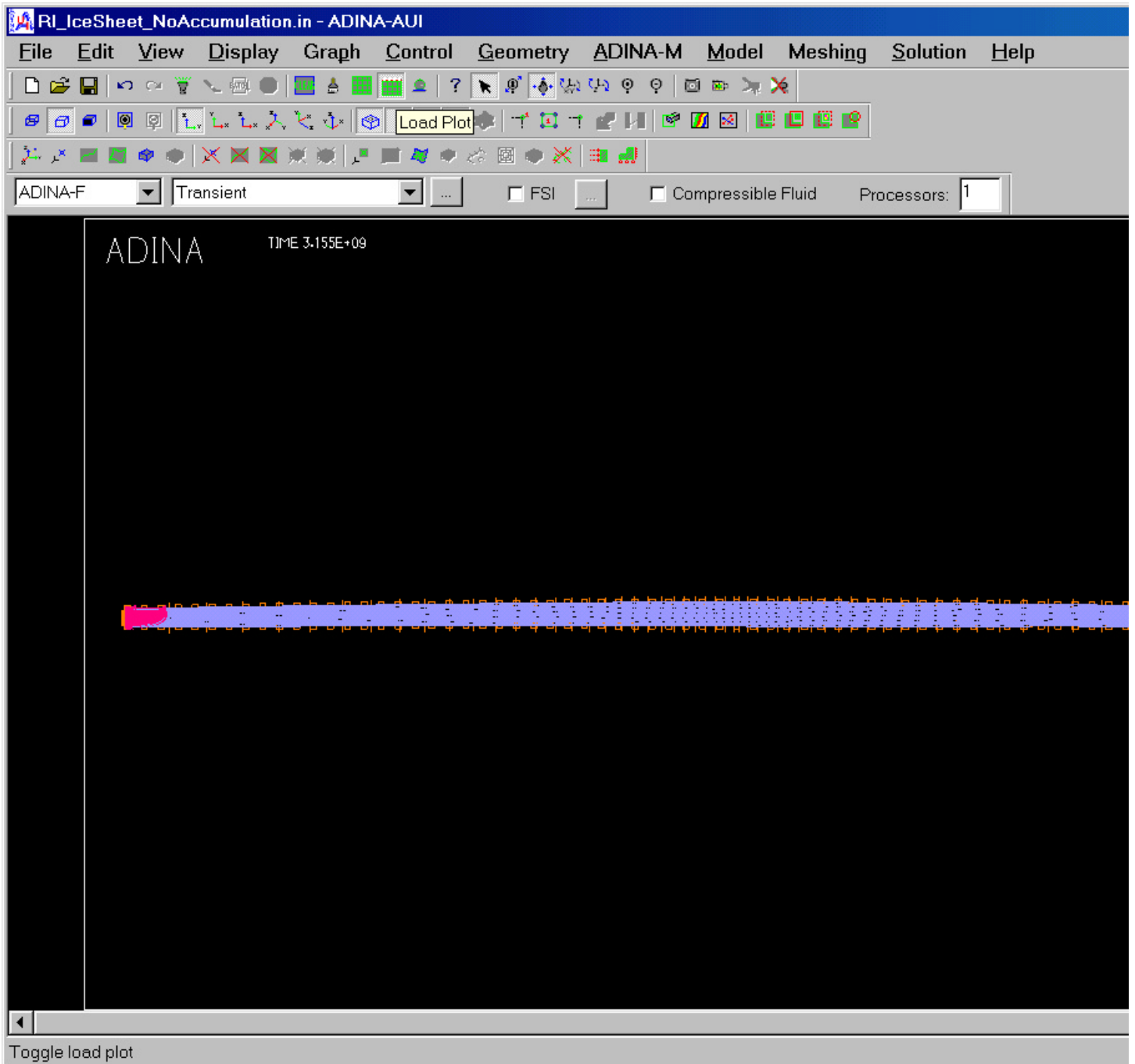
To give it a try, start up ADINA-AUI and load this file: [RI IceSheet NoAccumulation.in](#)

Start up ADINA, then Menu Option: **FILE->OPEN**

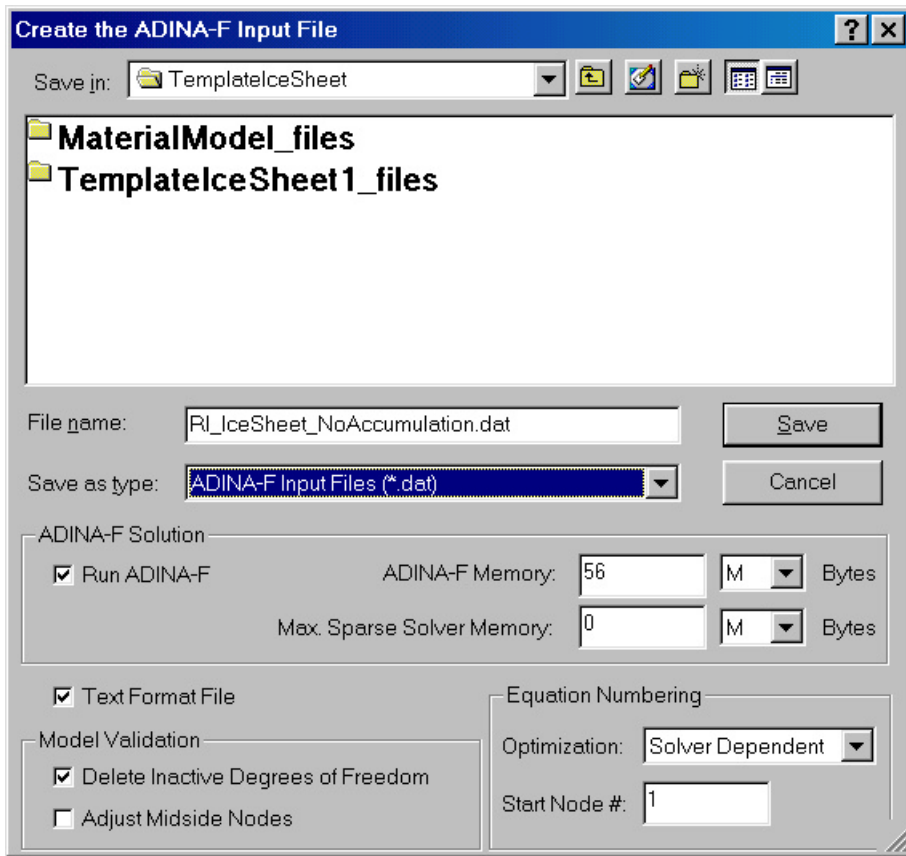
make sure to change file type to "ADINA-IN Command Files" as below.



Then do a load plot to see the model setup. This is accomplished by clicking on the icon shown in this image:

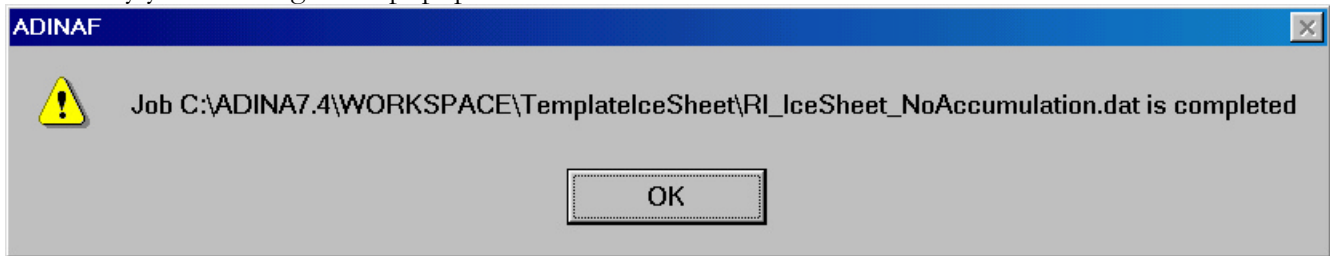


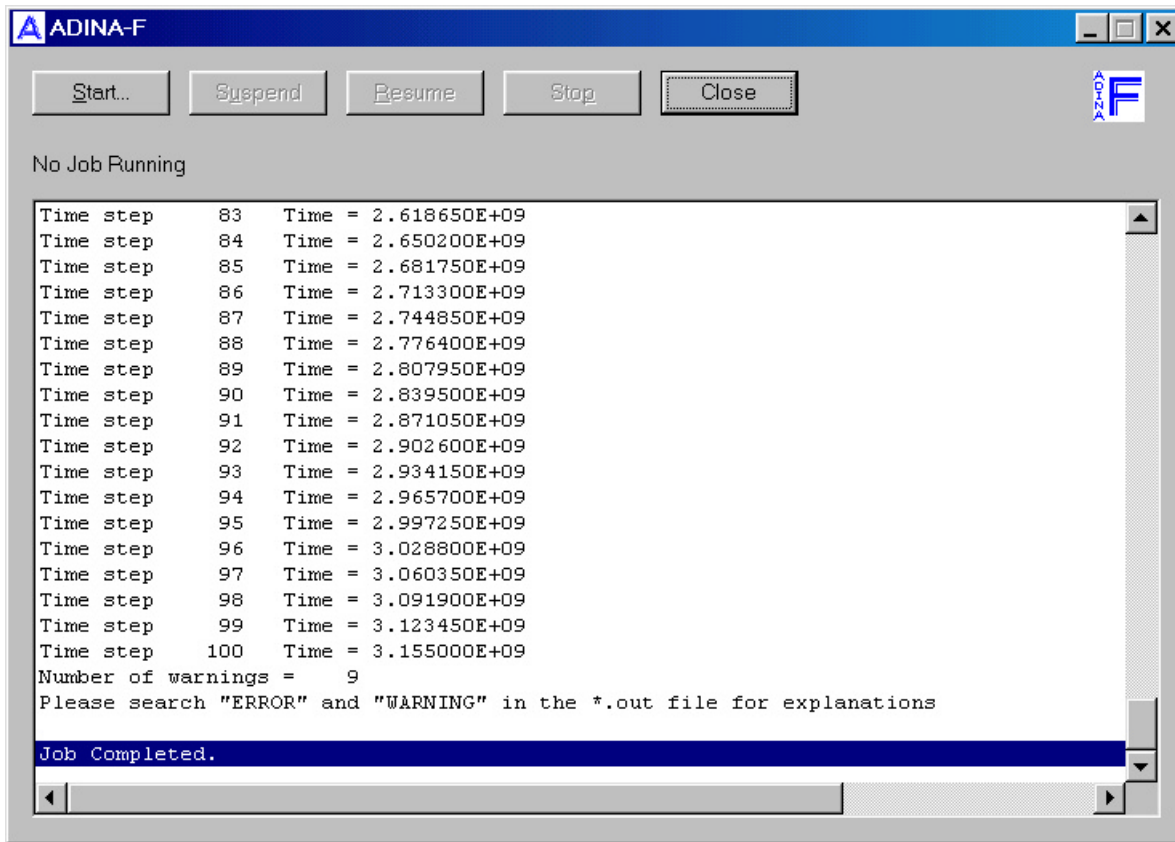
Now run the thing. Menu Option: **SOLUTION->DATA FILE/RUN**
You have to give the output ".dat" file a name.



The initial time steps will bring up some warnings - this is the result of the model not being stable with the initial setup. This settles out after a few time steps.

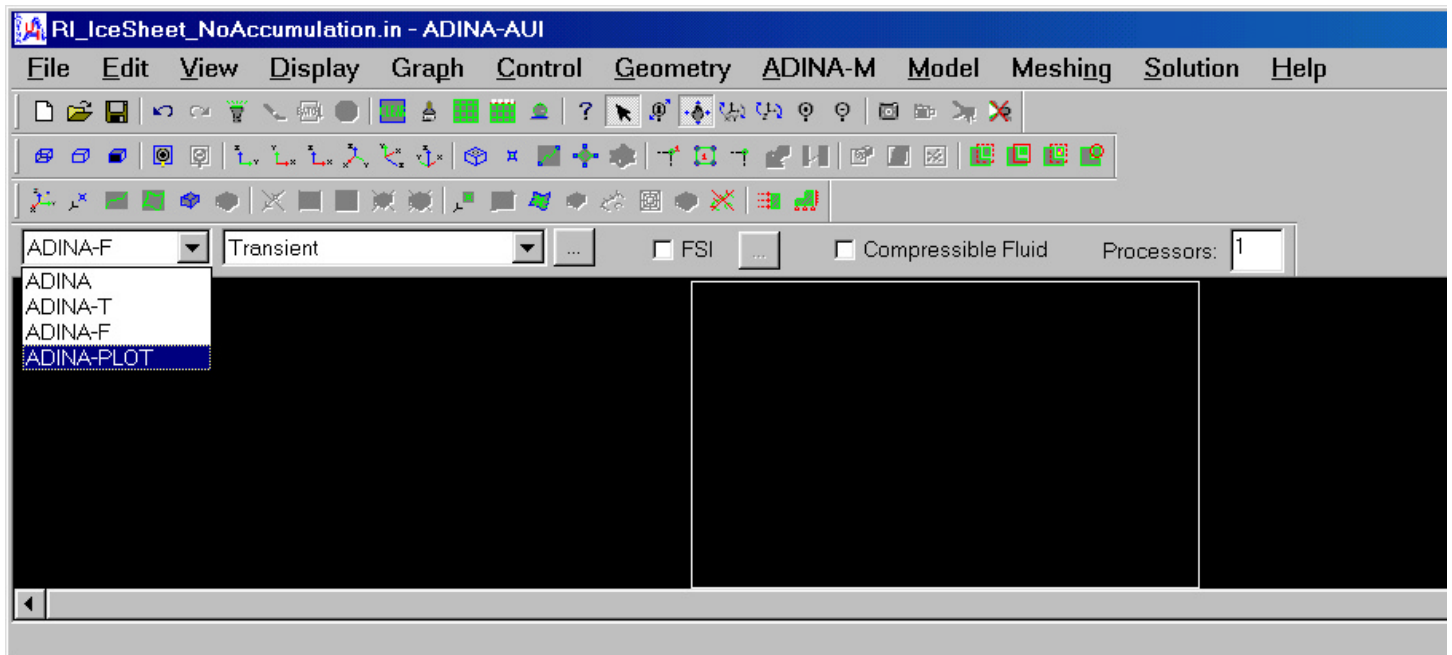
Eventually you should get this popup box:



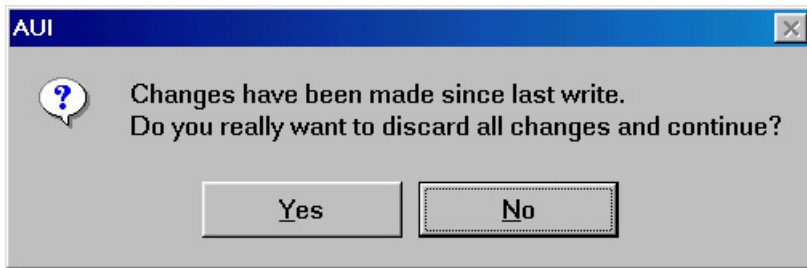


14) EXAMINE THE RESULTS

We have to switch programs to ADINA-PLOT.



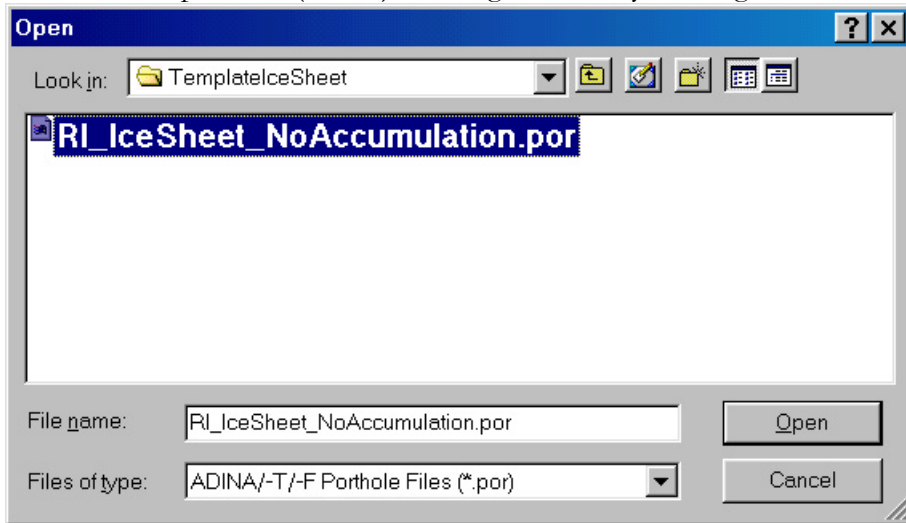
Click yes to the question:



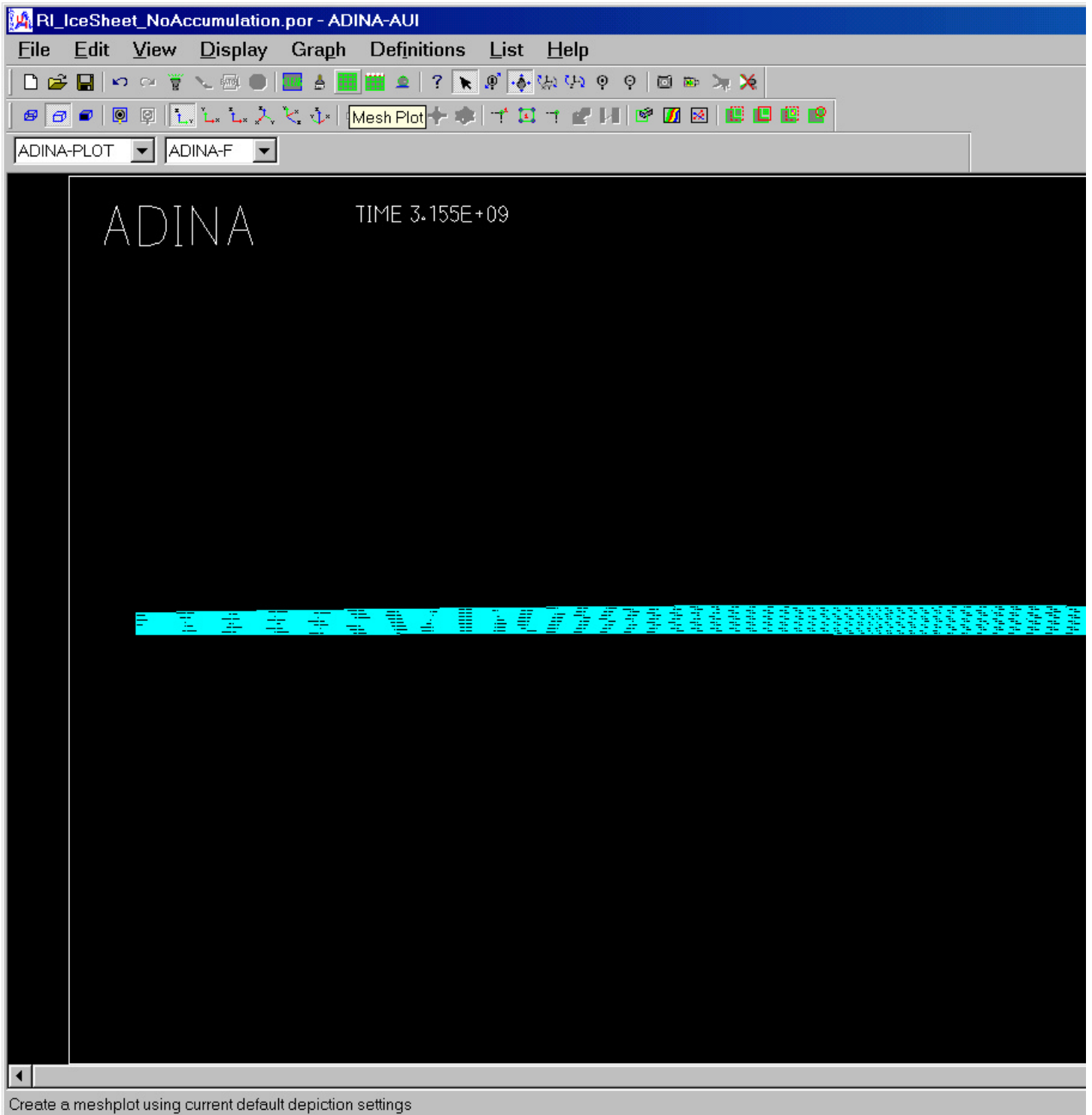
Now we need to load the results file.

Menu Option: FILE ->OPEN

and select the porthole (results) file we generated by running the model.

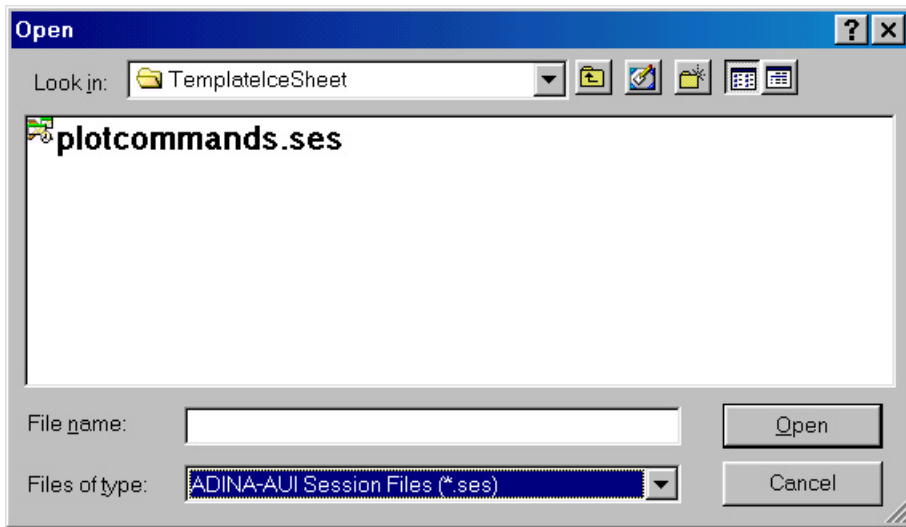


It is good to do a mesh plot which shows the results and the deformed mesh:

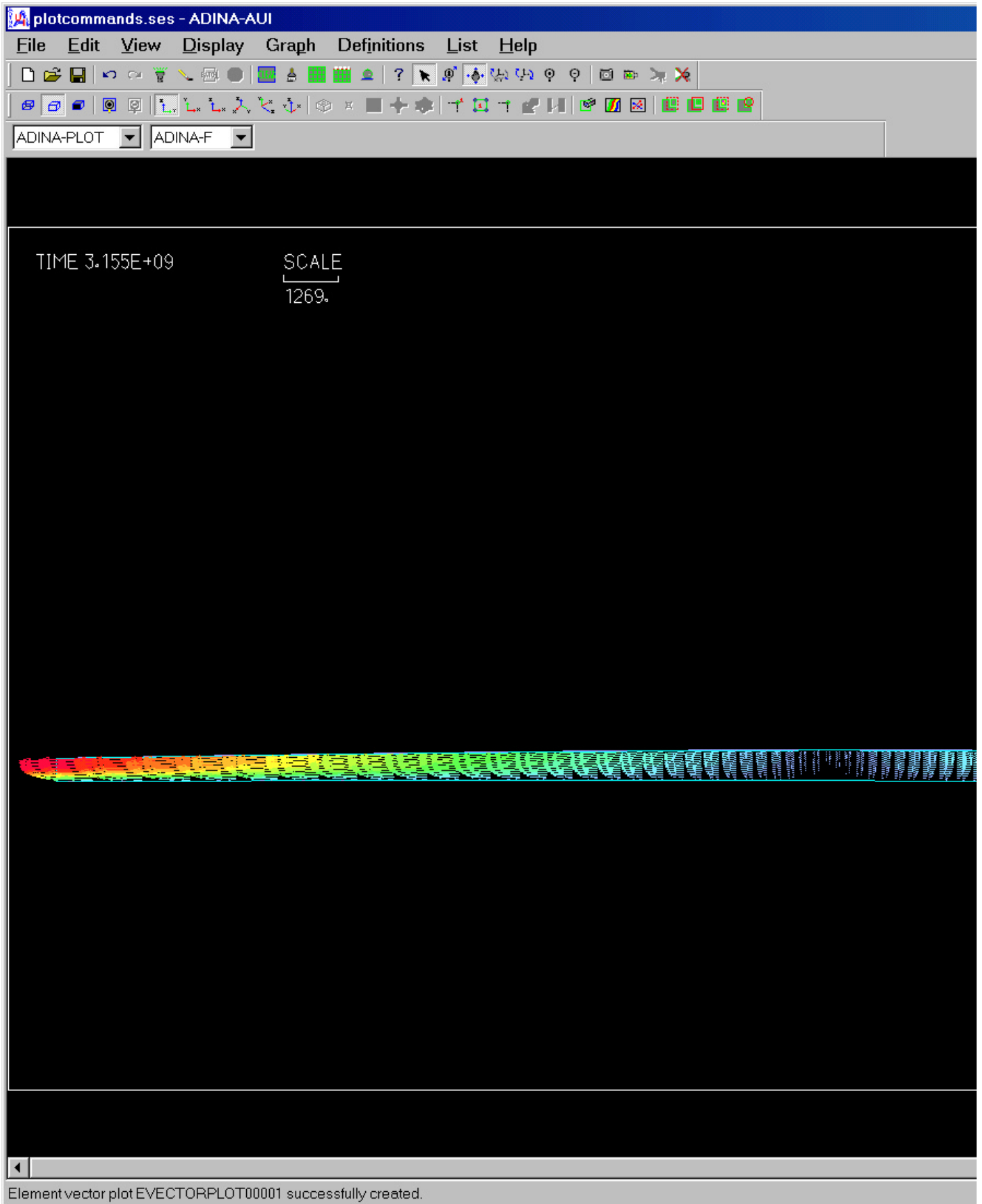


But, the best way to view the results is to get rid of the mesh. There is a standard set of settings that I make to the plotting options. I have included in this file: [plotcommands.ses](#)

This file sets up the plotting preferences to only plot the outline of the model. Load this sequence of commands with MENU OPTION: FILE ->OPEN then select file type of "Session Files" and load the plotcommands.ses file. The commands will automatically run.



This will yield an image something like this - a vector plot showing the end velocity field. In the plotcommands file, I have selected that the velocity vectors be scaled to yield meters/year rather than the useless meters/second.



After the

