FIDAP Tutorial 3: Curtain Coating

Introduction: The aim of this tutorial is to set up and solve a problem involving curtain coating. There are a number of different industrial processes for covering a substrate with a thin layer of coating material such as the manufacturing of photographic film. Three of the common methods include curtain coating, slot coating and slide coating. In curtain coating, the liquid freely falls over a distance before it is layered over a moving substrate. Two of the advantages of curtain coating is that even very irregular shaped substrates can be covered with a thin layer of liquid and coating can be achieved at very high speeds. The flow in curtain coating can generally be divided into three zones: sheet forming zone, curtain flow zone, and impingement zone (see Figure 3-1). The impingement zone generally is the most important to understand because it ultimately controls the entire process. Of primary importance are the free surface thickness and the heel formation. These depend on the physical properties and the ratio of the speed of the falling liquid to the speed of the moving substrate. Heel formation is important to minimize in order to avoid the formation of eddies. Eddies may trap bubbles and lead to uneven coating. They are also susceptible to flow instabilities. This tutorial demonstrates how to model the impingement zone of the curtain coating application.

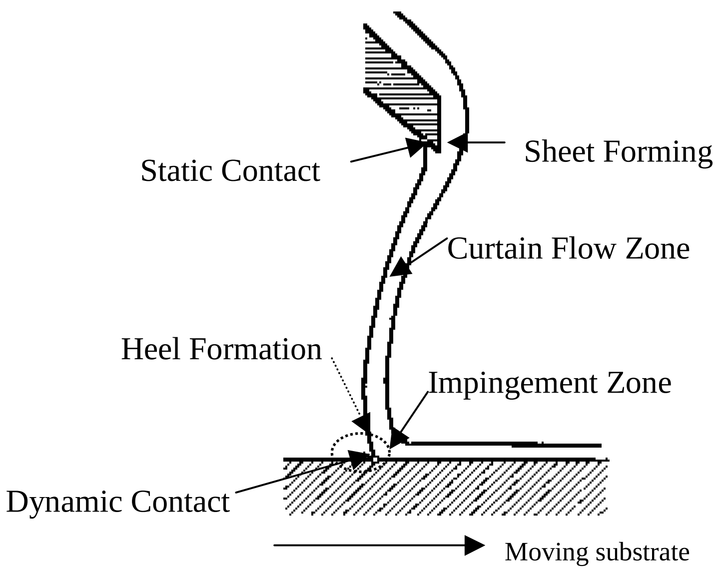


Figure 3-1: Curtain Coating

Objective:

- Solve a problem involving multiple free surfaces which arises in Curtain Coating Applications.

Prerequisites: This tutorial assumes that you have already completed FIDAP Tutorial 1: Laminar Flow in a 2-D Mixing Elbow.

For a review of Free Surface Simulations in FIDAP, refer to

- FIDAP Tutorial Manual: Chapter 6,
- FIDAP Theory Manual: Chapter 9,
- Introductory FIDAP Lecture Notes: Lecture 7

A one day Advanced Course in Free Surface Simulations is also available. Dates are available at www.fluent.com.

Files Needed for Tutorial 3:

These files can be downloaded from www.fluent.users.com/fidap8/training

- ccoat.FIPREP (Note: This file was modified by removing the "/" preceding the commands for DENSITY and VISCOSITY. This step will be unnecessary to do in .FIPREP files created by GAMBIT versions 2.x and later.)
- ccoat.FDNEUT

Problem Description: An approximation to the final free surface shape is shown schematically below in Figure 3-2. Cgs units are used and the units of length are given in cm. The curtain thickness is .06 cm and falls from .48 cm above the moving substrate. In this example the speed of the falling liquid curtain at the "inlet" and the moving substrate "slip" are equal. Conservation of mass requires that the flow rate at the "inlet" and "outlet" match. A plug flow velocity profile (UY=-200 cm/s) is prescribed at the "inlet". There are two free surfaces. The left free surface is "lfree" and results in a dynamic contact line. Typically high stresses develop near the dynamic contact point causing numerical oscillations which prevent convergence. A work around is to use the Navier slip condition for a few of the nodes on the solid boundary nearest the dynamic contact line. For this purpose the solid boundary needs to have the entity type SLIP. For greater control of the mesh, the domain is divided into two faces (separated by the dashed line in the figure below) and the corresponding two edges which comprise the right free surface are "rfree1" and "rfree2". In this case, we first compute the Reynolds number which is a ratio of inertial and viscous forces. At the "outlet" the Reynolds number (Re= 1.25 * 200 * .06/5.27) is 2.85 which puts the flow in the laminar regime.

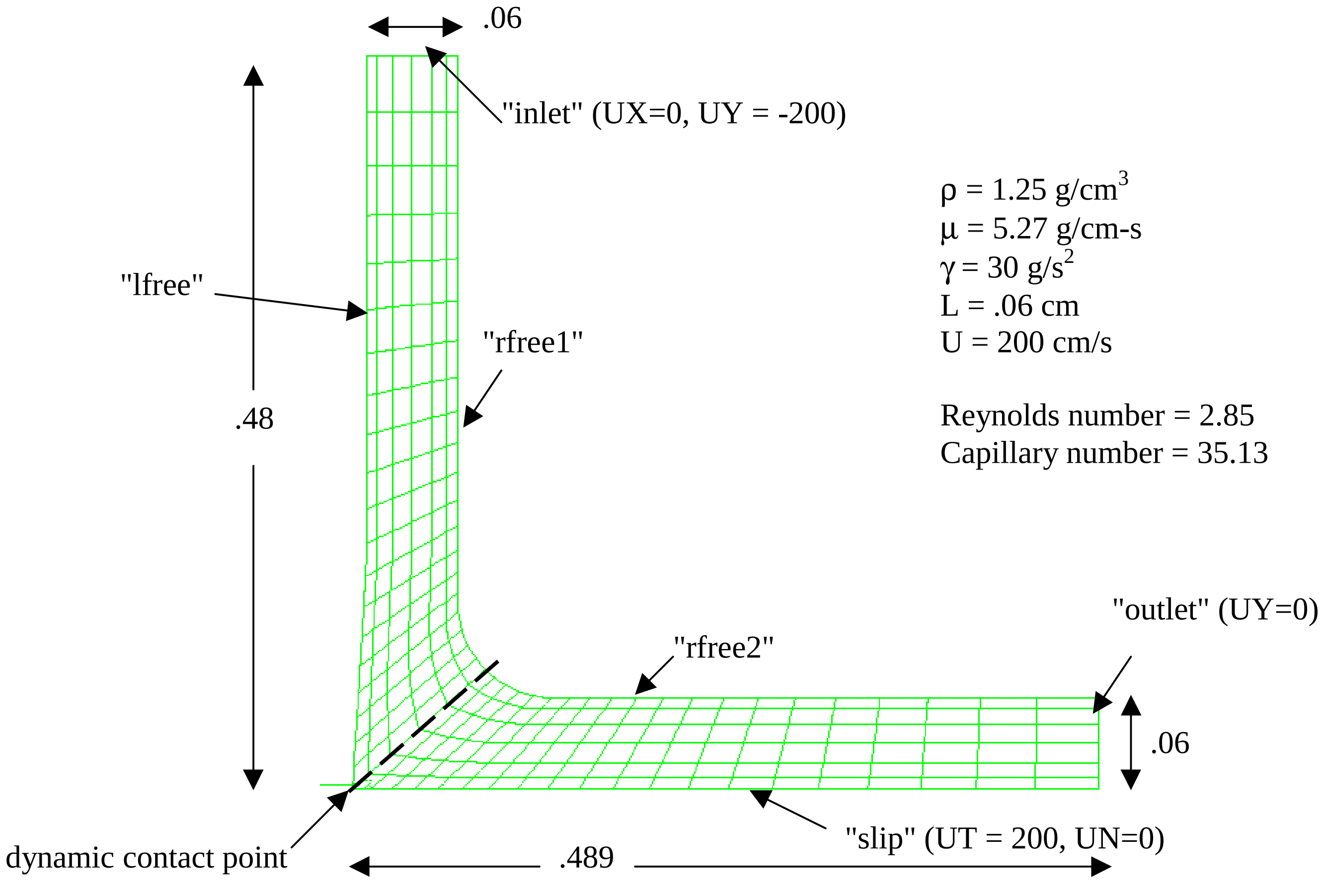


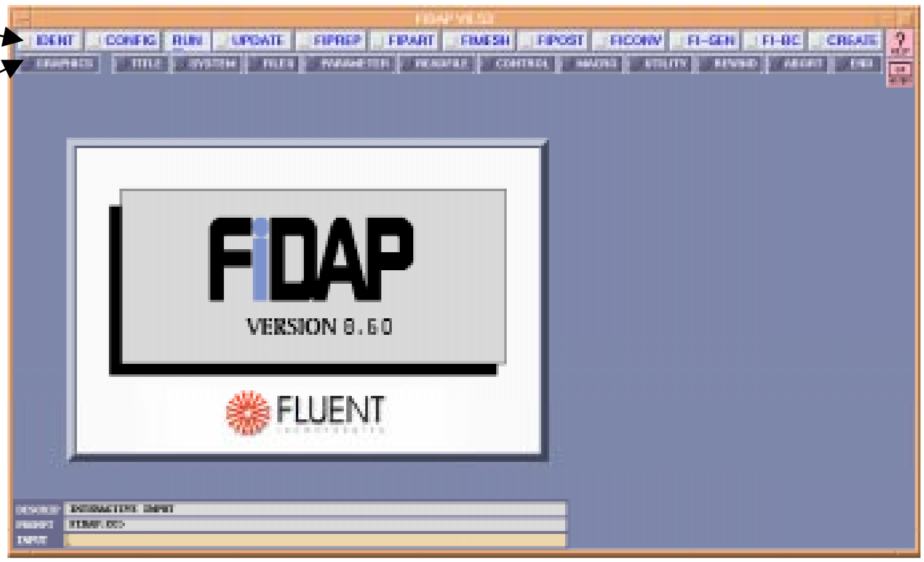
Figure 3-2: Mesh Plot

STEP 1: Start up FIDAP

Start up FIDAP in the GUI mode using the identifier: ccoat

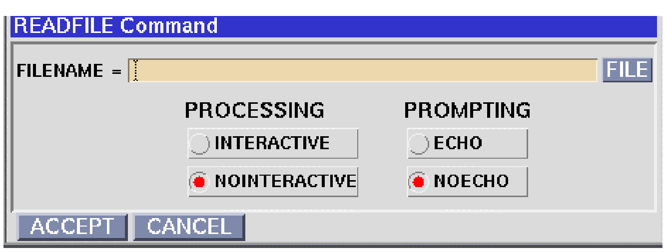
fidap –id ccoat –gui

Main Module Toolbar
FIDAP Command Toolbar



STEP 2: Read in the .FIPREP file

1) *The commands for the preliminary problem description have been created by the mesh generator (GAMBIT) and are contained in the file ccoat.FIPREP (see Appendix A). You will save a few steps by reading in this file.* Select the READFILE option on the FIDAP Command Toolbar near the top center of the FIDAP screen (Do **NOT** click FIPREP on the Main Module Toolbar). The READFILE Command form is shown below.



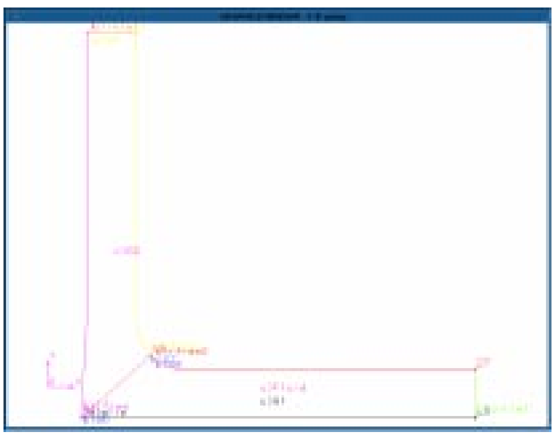
2) Click FILE on the READFILE Command form. The Select Read File Form appears. Highlight the file: ccoat.FIPREP and click ACCEPT at the bottom of the Select Read File form. Click ACCEPT at the bottom of the READFILE

Command Form. After successfully processing this file, the following message will appear in the Command History Window:

```
*****FISOLV INPUT DATA SUCCESSFULLY CREATED
```

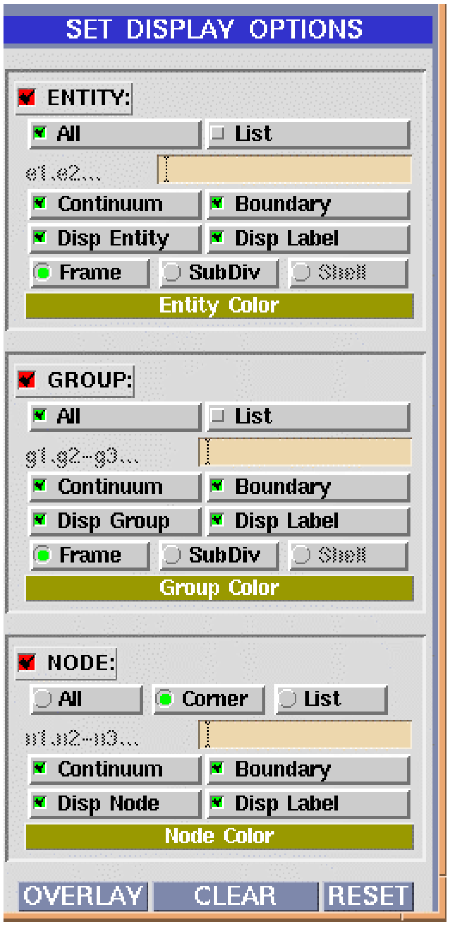
STEP 3: Enter the FI-BC module

1) Click FI-BC on the Main Module Toolbar. The entity names (defined in GAMBIT) and group numbers are displayed in the graphics window. For example, the entity "inlet" is also Group 3 (G3) and it is located near the top left of the graphics window. There are two distinct free surfaces. The left free surface is "lfree". The right free surface is composed to two different entities: "rfree1" and "rfree2". The fluid enters at "inlet" (top) and exits at "outlet" (right). The horizontal edge is "slip". The continuum entity is "fluid".

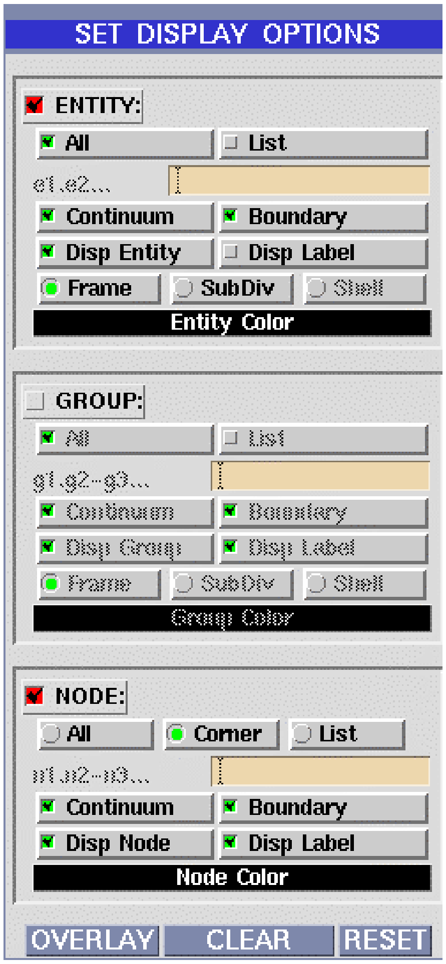


The FI-BC Command Toolbar is shown below.

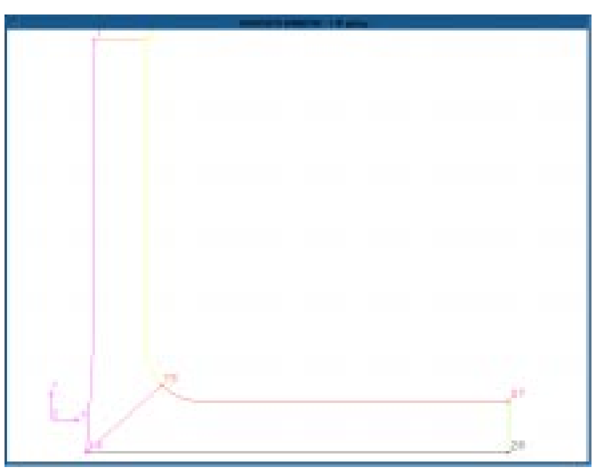
2) Click Set Display on the FI-BC menu toolbar. The SET DISPLAY OPTIONS Command Form is shown below.



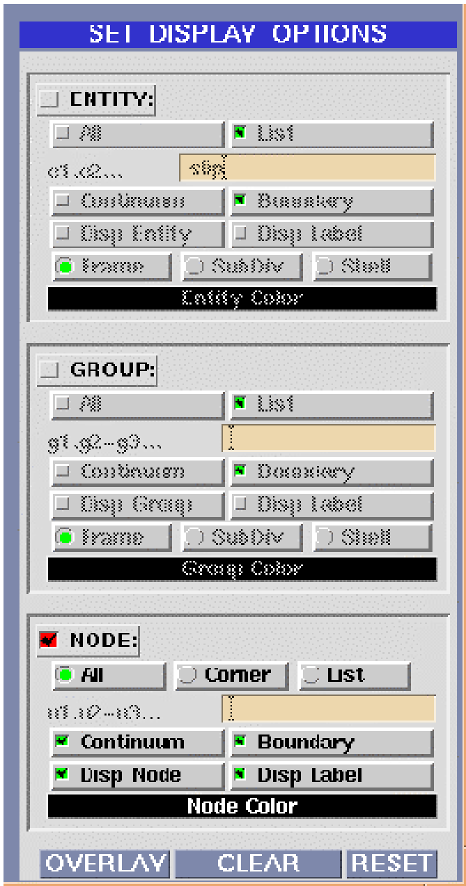
3) Notice that the form is divided into three sections: ENTITY, GROUP and NODE. Click off Disp Label in section 1: ENTITY. Click off GROUP. The updated SET DISPLAY OPTIONS Command Form is shown below.



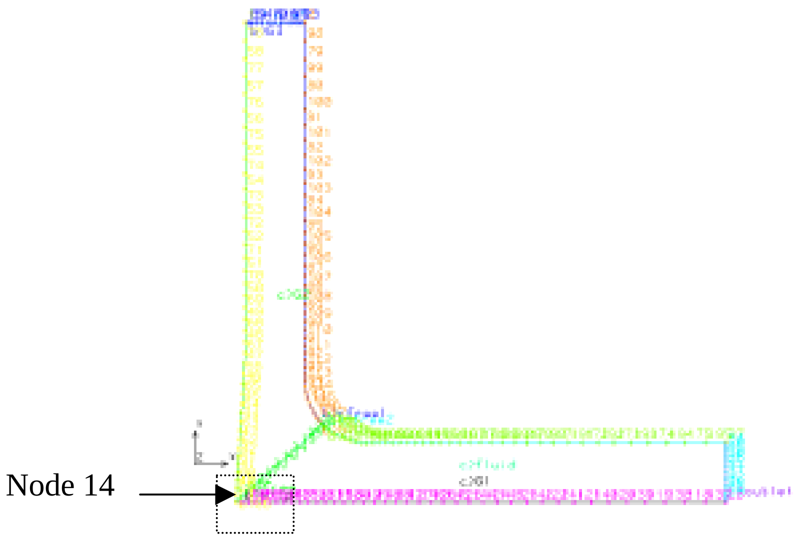
4) Click CLEAR at the bottom of the SET DISPLAY OPTIONS Command form. The edges and vertex node numbers are displayed in the graphics window. The corner node numbers are 1, 2, 27, 28 and 14. *These node numbers will be used in Step 15 #11 - #12 in order to define a physically correct normal at these corner points. If you created your own mesh, the node numbers will most likely differ from the ones shown below.*



5) *It is also necessary to free the horizontal velocity component of the 3 closest nodes to node 14 that lie on the entity "slip". This step allows the user to determine those node numbers*. Click off ENTITY. Click ALL in section 3: NODE. The updated form is shown below.

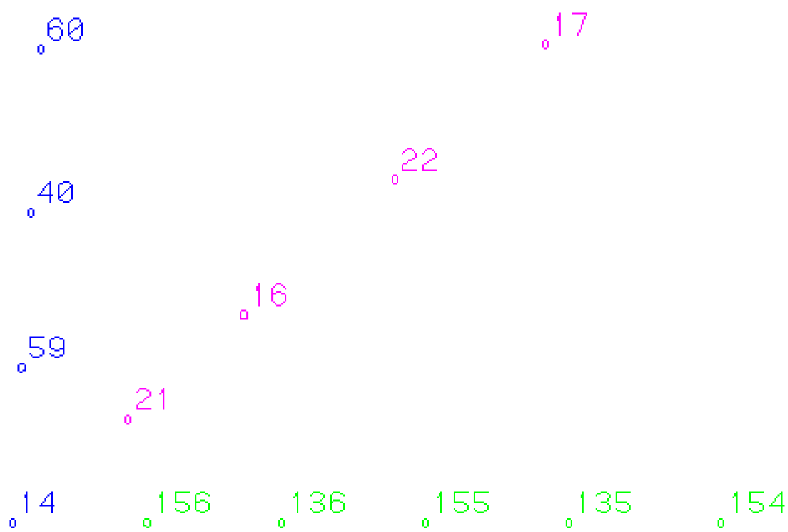


6) Click CLEAR at the bottom of the form. All of the node numbers are displayed.



Zoom into the region next to node 14 by moving the cursor to the region just below and to the left of node 14. Press the C key, depress the left mouse button and drag a small rectangle around the 3 nodes just to the right of node 14. Release the left mouse button.

7) The 3 nodes just to the right of node 14 are now easily visible as shown below. They are node numbers: 156, 136 and 155.



8) Click END on the FIDAP toolbar to end out of the FI-BC module.

STEP 4: Enter the FIPREP module

Click FIPREP on the Main Module Toolbar. The FIPREP Command Toolbar is shown below.

At this stage, the only information in the FIDAP database has been provided by GAMBIT from the .FIPREP file (see APPENDIX A). The .FIPREP file created by GAMBIT assumes certain basic commands and they may or may not be appropriate. In this example, the problem statement assumes that no free surfaces are present and the physical properties (density and viscosity) are both equal to one. Clearly, these two commands need to be changed. Table 3-1 describes which FIDAP commands need to be modified.

Table 3-1: Commands that need to be Modified

Command	**Current FIDAP database**	**FIDAP commands which need modification**
1. PROBLEM	2-D, LAMINAR, ISOTHERMAL, NONLINEAR, FIXED	Change FIXED to FREE
2. DENSITY	CONSTANT = 1	CONSTANT = 1.25
3. VISCOSITY	CONSTANT = 1	CONSTANT = 5.27
4. ENTITY	Only NAME and TYPE for boundary entities	SURFACE types need information on free surface movement. SLIP type needs PROPERTY set.

*In the .FIPREP file there are **NO** commands for the solver type, pressure algorithm, boundary or initial conditions. **IF NONE ARE ADDED BY THE USER, FIDAP WILL USE THE DEFAULT VALUES WHICH MAY OR MAY NOT BE APPROPRIATE.** Table 3-2 describes which FIDAP commands need to be added.*

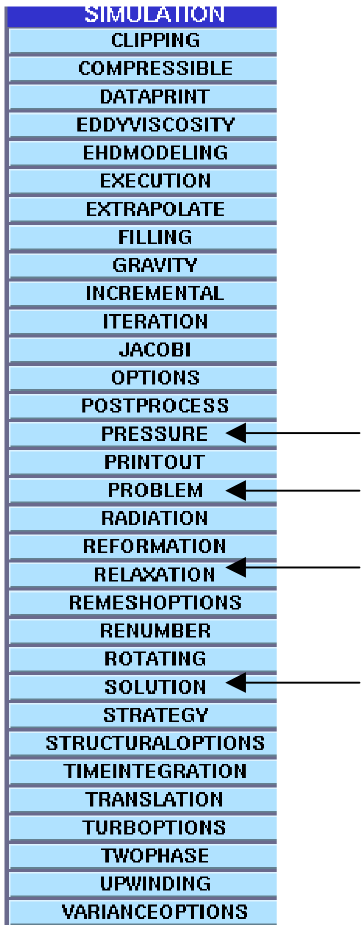
Table 3-2: Commands that need to be Added

No Command	Current FIDAP database (contains FIDAP default)	FIDAP commands needing to be added
1. Surface Tension	SURFACETENSION = 0	SURFACETENSION = 30
2. Solver	S.S. = 10	SEGREGATED
3. Pressure	Penalty (when solver is S.S.)	MIXED and DISCONTINUOS
4. BoundaryConditions	Zero normal stress	Boundary conditions needed
5. Initial Conditions	All D.O.F. = Zero	UY = -200
6. Scale	Value = 1	Value = .06
7. Relaxation	No relaxation: ACCF = 0 (when solver is S.S.)	Hybrid with user defined values

It is important to note that the defaults for pressure and relaxation are different for the S.S. and the SEGREGATED solver. Table 3-2 lists the default for the S.S. solver: Penalty method for pressure and No relaxation. If the solver is changed to SEGREGATED, the defaults change: Mixed Discontinuous for pressure and Hybrid Relaxation.

STEP 5: Define the PROBLEM Command

1) Click Simulation on the FIPREP Toolbar. The SIMULATION Toolbar is shown below.



Of the SIMULATION commands listed above, the following four commands will either need to be modified (if they already exist in the FIDAP database) or they will need to be added (if they do not exist in the FIDAP database):

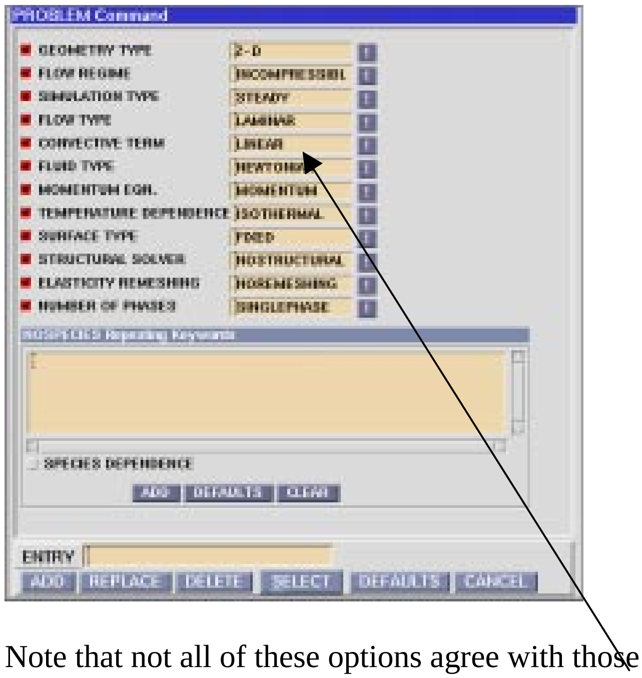
a) PROBLEM (modify by changing fixed to free),

b) SOLUTION (add the segregated solver instead of using the default S.S. solver),

c) PRESSURE (add: optional step since by changing the solver from S.S. to SEGREGATED, FIDAP will automatically change the default PRESSURE formulation to MIXED DISCONTINUOUS)

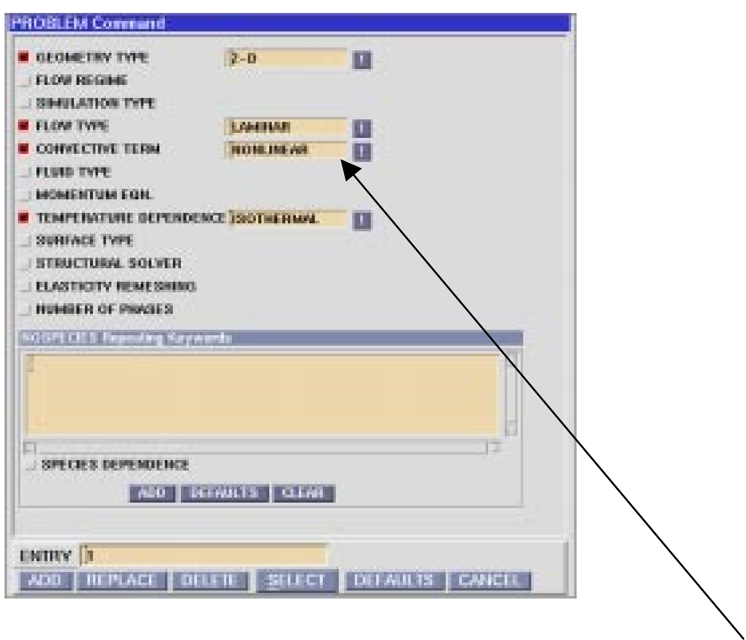
d) RELAXATION (add and specify relaxation values for all degrees of freedom)

2) Click PROBLEM on the Simulation Toolbar. The PROBLEM Command Form is shown below.



Note that not all of these options agree with those given in Table 3-1. The GUI shows that the CONVECTIVE TERM is LINEAR. An additional step is required to update the GUI to show the contents of the current database.

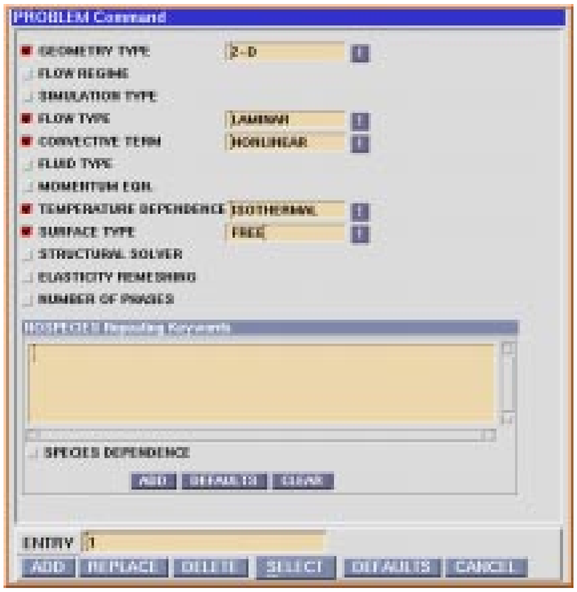
3) Click SELECT at the bottom of the PROBLEM Command and choose ALLENTRIES. The updated PROBLEM Command is shown below.



The form now shows the CONVECTIVE TERM to be NONLINEAR. All of these options are consistent with those shown in Table 3-1.

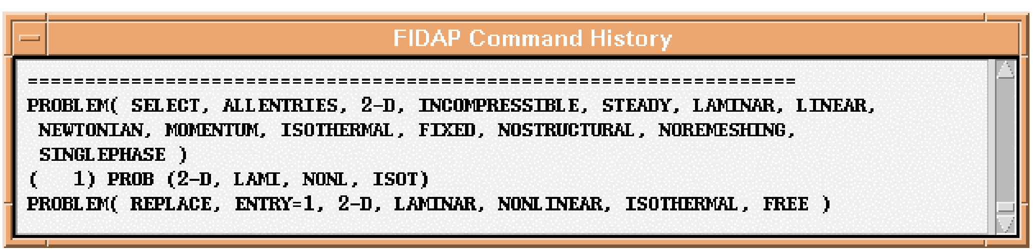
*The SURFACE TYPE is not selec*ted, which means that the default: FIXED simulation is active.

4) In the PROBLEM Command form, click SURFACE TYPE and right click the ! which appears to the right of FIXED. Change the keyword to FREE. The updated form is shown below:



5) Click REPLACE at the bottom of the PROBLEM Command Form. If you click ADD, there willl be two PROBLEM commands in the database. The last PROBLEM command will overwrite the previous one. This is not incorrect, it means there will be extra lines in the .FIJOUR file.

6) Confirm the change was made correctly by checking the FIDAP Command History Window.



STEP 6: Define the Solver

*The user must next decide on a solver. For free surface problems, there are two choices: the Newton Solvers and the Segregated Solvers A further consideration must also be given to the type of free surface update which determines how the nodes on the free surface are to move. The two choices are Kinematic and Normal. The kinematic update is generally preferred for problems in which the Capillary number >1 and the Normal stress update is generally preferred when the Capillary number < 1. The Capillary number is a ratio of viscous and surface tension forces (viscosity * velocity / surface tension = 5.27 * 200 / 30) and is given by 35.13. Table 3-3 shows which updates are available with each of the solvers.*

Table 3-3: Choice of Free Surface Updates For Newton and Segregated Solver

Newton Solver	**Segregated Solver**
Kinematic (Ca>1)	Kinematic (Ca >1)
	Normal Stress (Ca < 1)

For this problem, Ca >1 and therefore, the Kinematic approach is preferred. If the problem was a large 2-D problem or a 3-D problem we would be forced to use the Segregated Solver because of its lower memory requirement. However, this is a relatively small 2-D problem, so that either solver could be used.

Another choice that needs to be made is the Remeshing Scheme which determines how the nodes below the free surface move. There are three remeshing choices: Spines, Mapped and Elastic. Even though the remeshing scheme is not specified until Step 14, it is important to consider it in this step because not all the remeshing schemes are available with both solvers. A general recommendation to use the Spines approach for problems in which the mesh deformation is not large, otherwise either the Mapped or Elastic approach is needed. Table 3-4 shows which remeshing schemes are available with either solver.

Table 3-4: Choice of Remeshing Schemes for Newton and Segregated Solver

Newton Solver	**Segregated Solver**
Spines (small mesh deformation)	Spines (small mesh deformation)
	Mapped (moderate mesh deformation)
	Elastic (very large deformation)

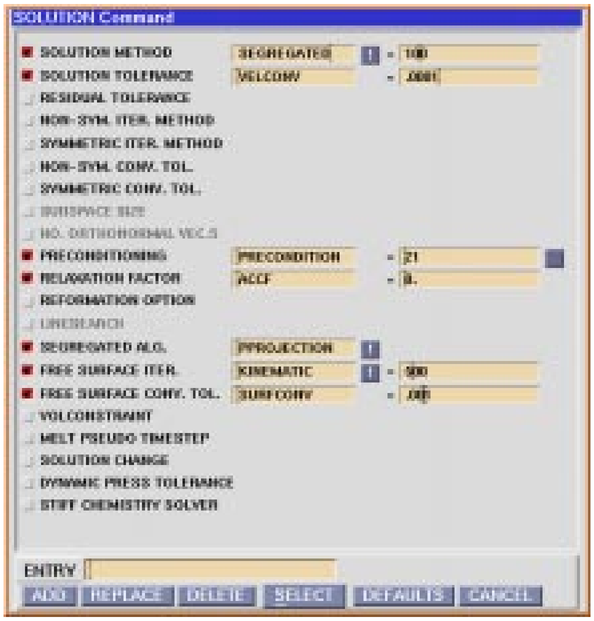
For this problem the mesh is expected to deform moderately, therefore the choice is the mapped scheme which is only available with the segregated solver.

Thus, using the Capillary number and estimation of mesh deformation, the choices are: the segregated solver and the mapped option for the free surface update.

The solver and free surface update are chosen in the SOLUTION command.

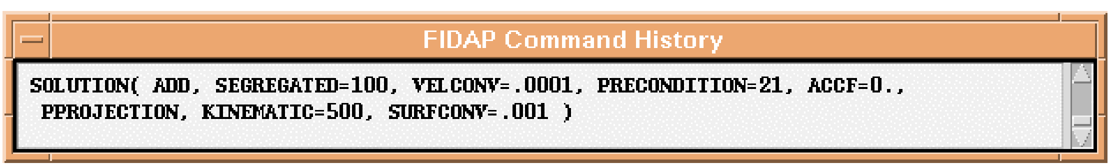
1) Click SOLUTION on the SIMULATION Command Form. The SOLUTION Command Form is shown below.

2) Right click on the ! to the right of S.S. and choose SEGREGATED. Change the value of the keyword SEGREGATED from 10 to 100. Click SOLUTION TOLERANCE and input the value .0001 (the default is .001) for the keyword VELCONV. Click FREE SURFACE ITER. and input a value of 500 for the keyword KINEMATIC. Click FREE SURFACE CONV. TOL and input a value of .001 (the default is .01) for the keyword SURFCONV. The updated SOLUTION Command form is shown below.



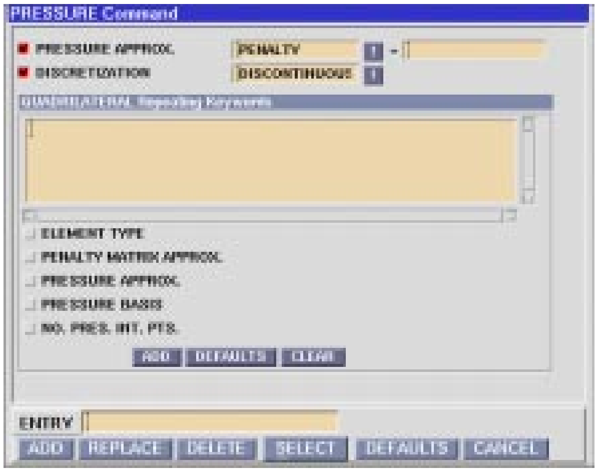
3) Click ADD at the bottom of the SOLUTION Command form.

4) Check the Command History Window to confirm the change was made correctly.

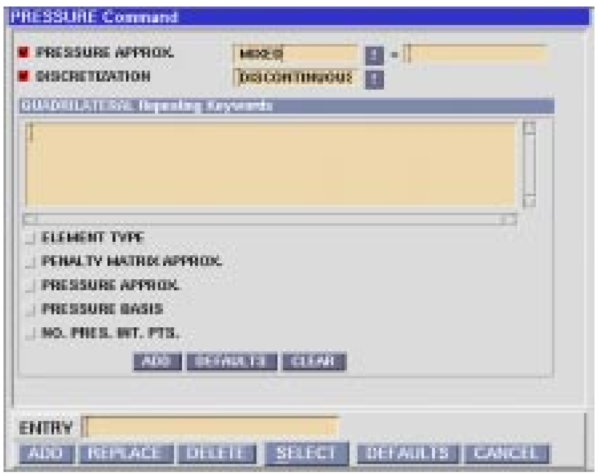


STEP 7: Define the PRESSURE Formulation

1) Click PRESSURE on the SIMULATION Command Form. The PRESSURE Command Form is shown below.



2) Change the PRESSURE APPROXIMATION from PENALTY to MIXED. The updated PRESSURE Command form is shown below.



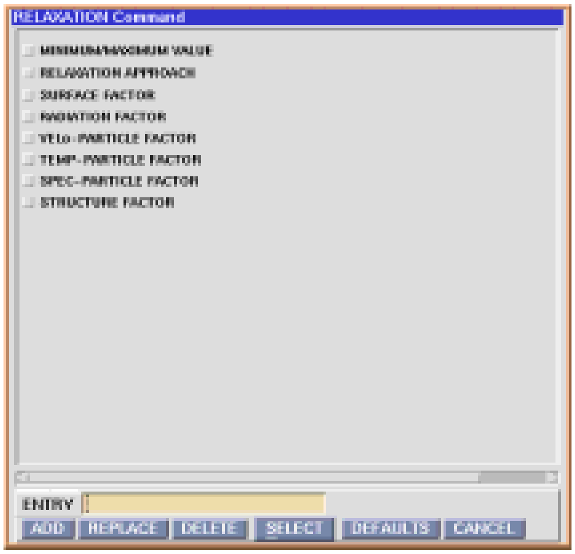
Note that there is no value given for the keyword MIXED, thus the default (1.e-6) will be used. This step is optional since the default pressure formulation is mixed, discontinuous whenever the segregated solver is used.

3) Click ADD at the bottom of the PRESSURE Command Form.

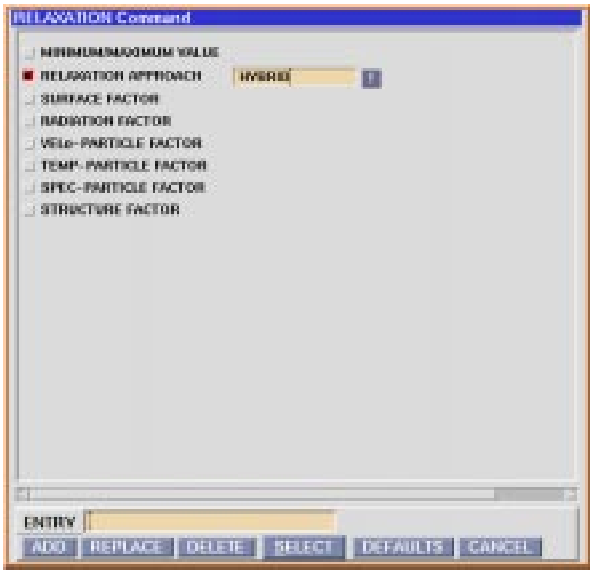
4) Check the Command History Window to confirm the change was made correctly.

STEP 8: Define the Relaxation Values

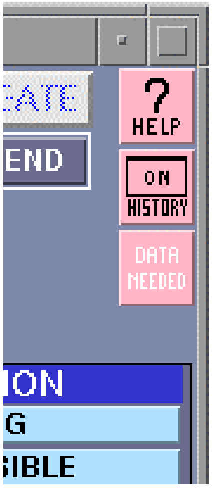
1) *Since the SEGREGATED SOLVER has been chosen, the default relaxation scheme is the HYBRID type. The default values for each degree of freedom are given in the FIPREP manual (.3 for velocity, .6 for pressure and .5 for free surface). In this step, the user will input a higher relaxation value (.975) for free surface.* Click RELAXATION on the Simulation Toolbar. The RELAXATION Command Form is shown below.



2) Click RELAXATION APPROACH and choose the HYBRID option. The updated RELAXATION Command form is shown below.

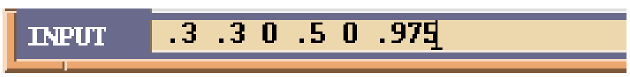


Click ADD at the bottom of the RELAXATION Command Form. The DATA NEEDED button will appear at the top right of the GUI.



3) You need to enter relaxation values for each of the degrees of freedom in the order:
UX, UY, UZ P , T and Free Surface

Move the cursor to the lower left wheat colored box and click in the box (or press the F2 key). Input the following data and then hit the Enter key:
.3 .3 0 .5 0 .975



The DATA NEEDED button will not disappear because FIDAP is expecting 22 values. Because only 6 values were entered, the remaining relaxation values will be given the default 0. Click on the DATA NEEDED button to make it disappear.

4) Check the Command History Window to confirm the change was made correctly.

STEP 9: Define the Physical Properties

The stress-divergence form of the momentum equation and the three boundary conditions are shown below. The user needs to input values for the properties: density (ρ), viscosity (μ) and surface tension (γ), otherwise the default value 1 is used.

$$\rho\left(\frac{\partial \mathbf{u}}{\partial t}+\mathbf{u}\cdot\nabla\mathbf{u}\right)=-\nabla p+\nabla\cdot\left[\mu\left(\nabla\mathbf{u}+\nabla\mathbf{u}^{T}\right)\right]$$

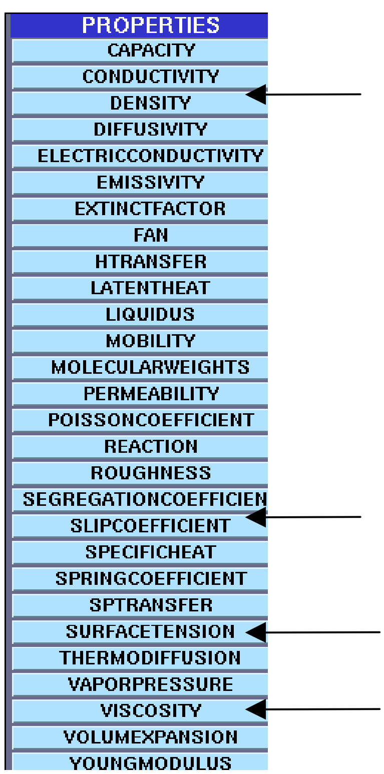
← Stress Divergence Form of Momentum Equation

$\mathbf{u}\cdot\mathbf{n}=0$	$\sigma_n=-p_a+2\gamma H$	$\sigma_t=\mathbf{t}\cdot\nabla\gamma$	← Boundary Conditions
Kinematic	Normal Stress	Tangential Stress	

In addition, the Navier Slip Condition is needed to alleviate the large local stresses near the dynamic contact point. The user needs to specify the slip coefficient a.

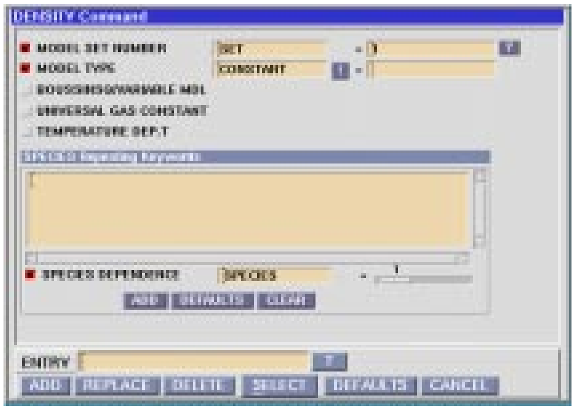
$$\sigma_t=\frac{1}{a}(\mathbf{u}-\mathbf{u}^{s})\cdot\mathbf{t}$$

Click Properties on the FIPREP Command Toolbar. The PROPERTIES Toolbar is shown below.

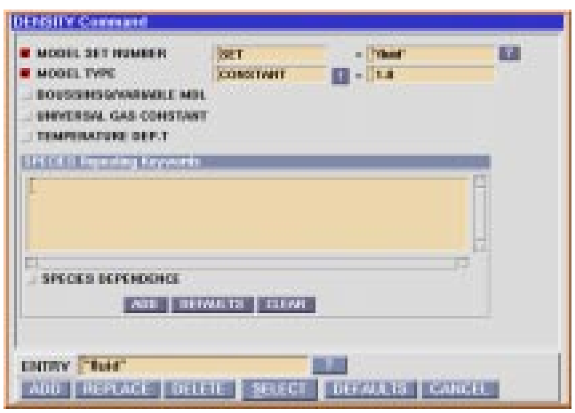


STEP 10: Define the DENSITY

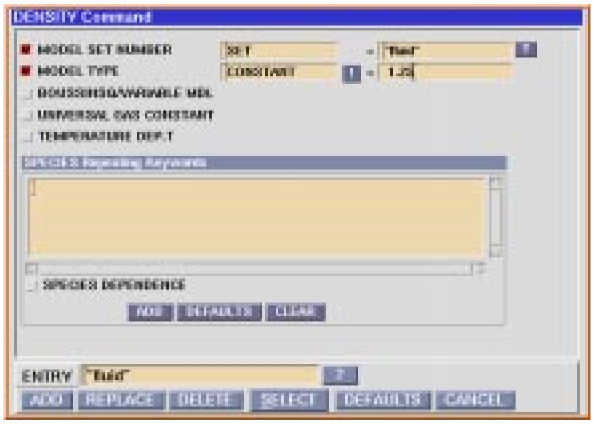
1) Click DENSITY on the PROPERTIES Toolbar. The DENSITY Command Form is shown below.



2) Click SELECT at the bottom of the DENSITY Command Form and choose ALLENTRIES. The updated DENSITY Command form is shown below. *It is important to note this requires that the DENSITY command in the .FIPREP file be uncommented. This is automatically done in GAMBIT versions 2.0 and later. If an earlier version of GAMBIT was used to create the geometry, the DENSITY command must be uncommented by the user. The same is true for the VISCOSITY command. This was done if you used the file provided with this tutorial (See the comment at the bottom of page 2: Files Needed for Tutorial 3).*

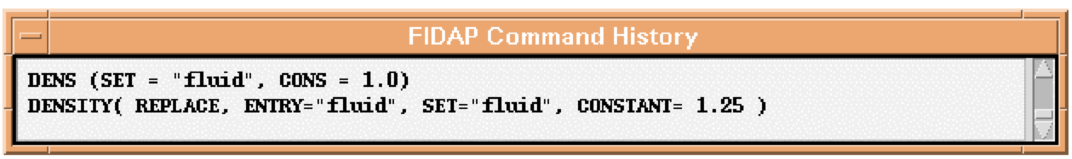


3) Change the value of the keyword CONSTANT = 1 to CONSTANT = 1.25. The updated DENSITY Command is shown below.



4) Click REPLACE at the bottom of the DENSITY Command Form.

5) Check the Command History Window to confirm the change was made correctly.



STEP 11: Define the VISCOSITY

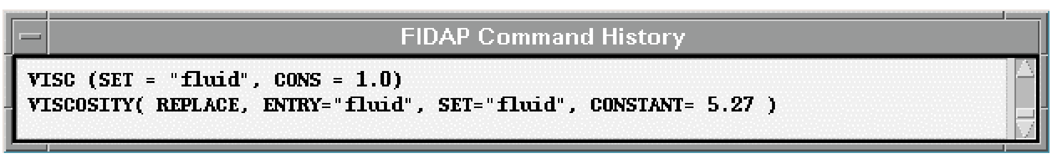
1) Click VISCOSITY on the PROPERTIES Toolbar. The VISCOSITY Command Form is shown below.

2) Click SELECT at the bottom of the VISCOSITY Command Form and choose ALLENTRIES. The updated VISCOSITY Command form is shown below.

3) Change the value of the keyword CONSTANT = 1 to CONSTANT = 5.27. The updated VISCOSITY Command is shown below.

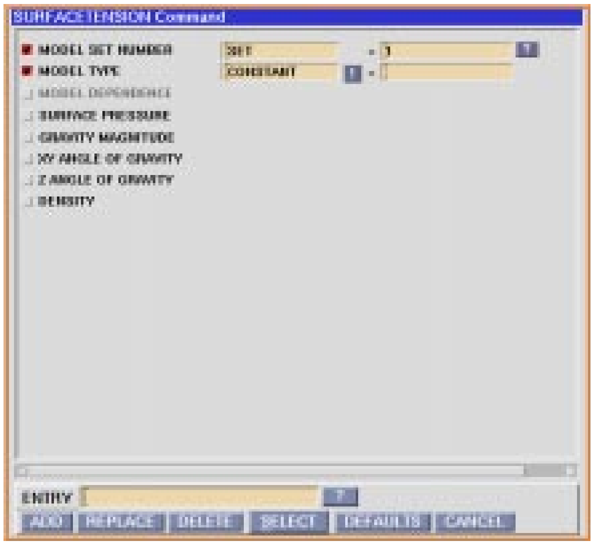
4) Click REPLACE at the bottom of the VISCOSITY Command Form.

5) Check the Command History Window to confirm the change was made correctly.

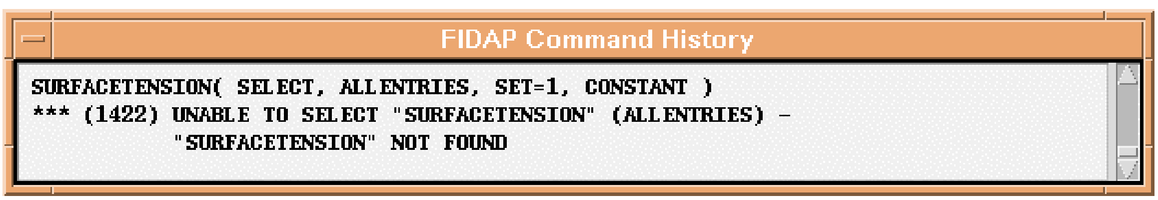


STEP 12: Define the SURFACE TENSION

1) Click SURFACETENSION on the PROPERTIES Toolbar. The SURFACETENSION Command Form is shown below.



2) Click SELECT at the bottom of the SURFACETENSION Command Form and choose ALLENTRIES. Note the warning message in the FIDAP Command History Window:

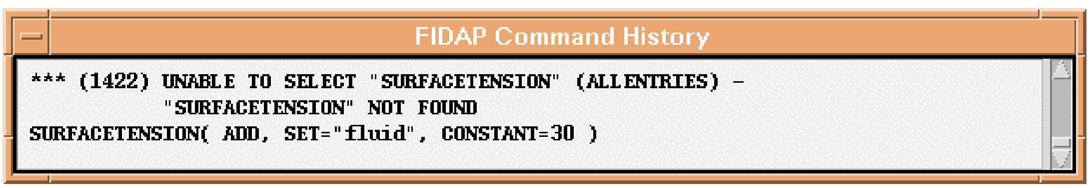


This warning error occurs because there are no SURFACETENSION Commands present in the current database.

3) Change the MODEL SET NUMBER from SET = 1 to SET = fluid. You do not need to type in the quotes. Input a value of 30 for the keyword CONSTANT. The updated form is shown below.

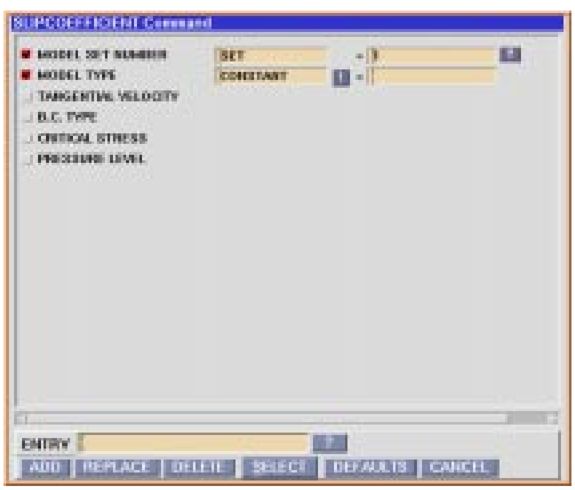
4) Click ADD at the bottom of the SURFACETENSION Command Form.

5) Check the FIDAP History Window to confirm the change was made correctly.

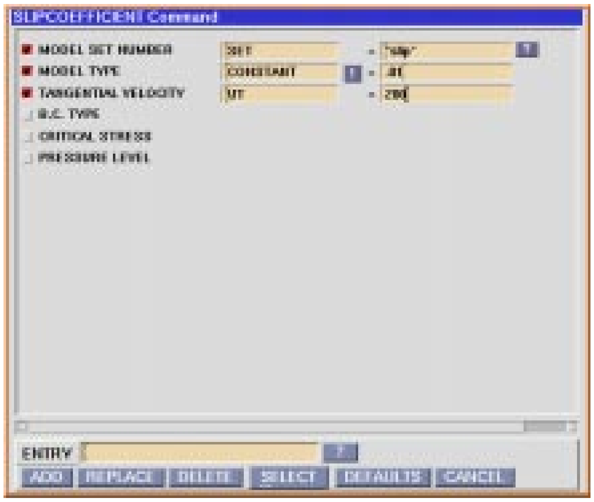


STEP 13: Define the SLIPCOEFFICIENT

1) Click SLIPCOEFFICIENT on the PROPERTIES Toolbar. The SLIPCOEFFICIENT Command Form is shown below.

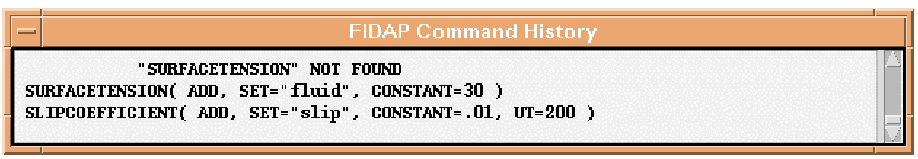


2) Change the MODEL SET NUMBER: SET = 1 to SET = slip. Input a value of .01 for the keyword CONSTANT. Click TANGENTIAL VELOCITY and input a value of 200 for the keyword UT. The updated form is shown below.



3) Click ADD at the bottom of the SLIPCOEFFICIENT Command Form.

4) Check the FIDAP History Window to confirm the change was made correctly.



STEP 14: Modify the ENTITY Commands

Several modifications need to be made to the ENTITY Commands for the SURFACE and SLIP types. The user needs to decide how the nodes on and below the SURFACE entities: "lfree", "rfree1" and "rfree2" are to move. The user also needs to determine the Slip Coefficient for the entity "slip".

It has already been decided that the remeshing scheme is the mapped option, which is determined on the ENTITY command under the option SPINE GENERATION.

The user additionally needs to decide

- *how many nodes below the free surface are allowed to move*
- *how the nodes on the free surface are to move*
- *the contact angle at the end points of all SURFACE entity types if the endpoints are not fixed (UX=UY=0).*

Specifying the DEPTH keyword determines how many nodes are allowed to move. The value is either

- *0 (maximum depth)*
- *-1 (no spines are created)*
- *positive number (specifies the depth including the node on the free surface).*

For the mapped approach the choices for determining how the nodes on the free surface move are

- *NORMAL (default)*
- *PREFERRED*

If the PREFERRED option is chosen the direction is specified by the values of the keywords PREFERRED DIRECTION -X, -Y and -Z.

The contact angle is chosen by specifying the angle the outward tangent of the free surface makes with the positive x- axis. It is only needed for nodes 14 and 27. (All of the other endpoints: nodes 1, 2, 28 are fixed. (Refer to the diagram shown in Step 3 #4 for the node numbers). For 2-D problems there are two options for specifying the contact angle.

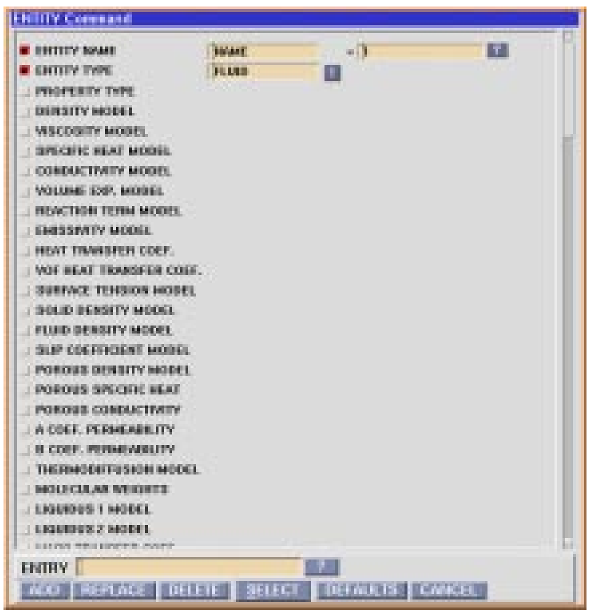
- *Use the ANG1 or ANG2 keyword on the ENTITY command.*
- *Use the BCNODE command, specify the DEGREE OF FREEDOM to be CONTACTANGLE and specify the NODE NUMBER at which the contact angle is being prescribed.*

In this tutorial we will use the first option to specify the contact angle at node 27 (Step 14 #8) and the second option to specify the contact angle at node 14 (Step 15, #15). An advantage of using the first option is that since the node number is not required, it would be the preferred choice if the geometry needs to be remeshed (which would change the node numbering). A summary of the correct modifications to the entity commands is given in Table 3-5.

Table 3-5: Modification For FREE Entity Types

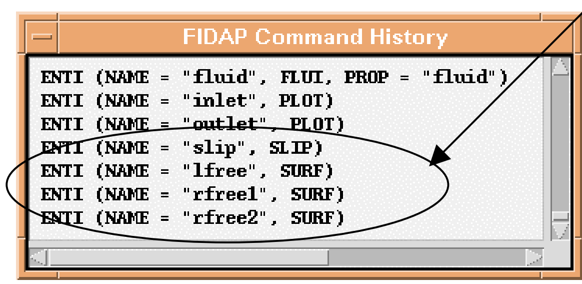
ENTITY	SPINE GENER-ATION	DEPTH	SURFACE DEFORMATION	Contact Angle 1	Contact Angle 2
"lfree"	MAPPED	0	PREFERRED, X=1	Not required (Node = 1)	Use BCNODE (Node = 14)
"rfree1"	MAPPED	-1	NORMAL (default)	Not required (Node = 15)	Not required (Node = 2)
"rfree2"	MAPPED	0	NORMAL (default)	ANG1 = 0 (Node = 27)	Not required (Node = 15)

1) Click Entity on the FIPREP Toolbar. The ENTITY Command Form is shown below.



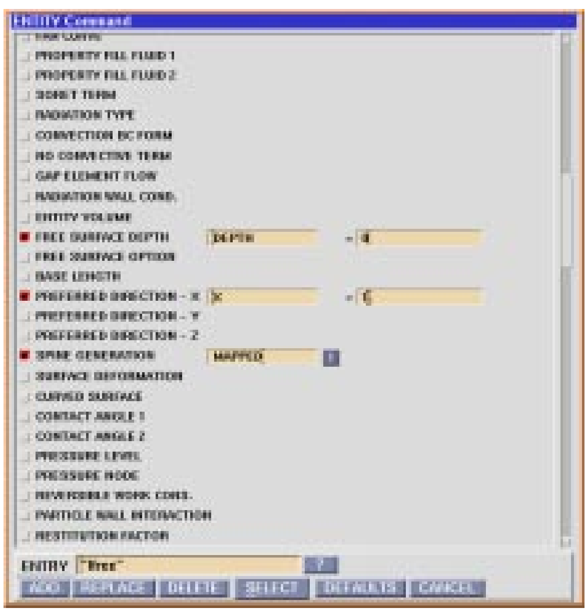
2) Click SELECT and choose ALLENTRIES. The updated form is shown below.

The FIDAP Command History Window lists all of the ENTITY commands in the current database. Only the commands for the last four entities: "slip", "lfree", "rfree1" and "rfree2" need to be modified.



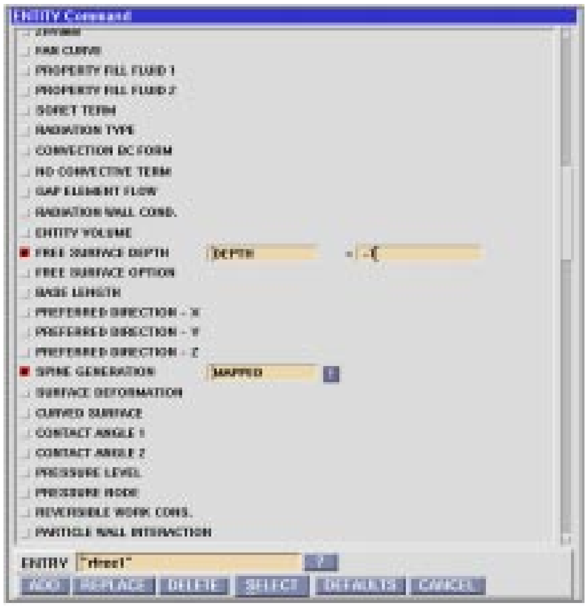
3) To modify the entity "lfree": Click the ? at the bottom of the form and choose "lfree". The updated form is shown below.

4) Use the vertical slider bar in the Command Form and scroll down one third of the way until you see the FREE SURFACE DEPTH. Click FREE SURFACE DEPTH and input a value of 0 for the keyword DEPTH. Click PREFERRED DIRECTION -X and input a value of 1 for the keyword X. Click SPINE GENERATION and choose the MAPPED option. The updated form is shown below.



5) Move the cursor into the grey area, right click and choose REPLACE(REPEAT). Confirm the change was made correctly by checking the FIDAP Command History Window.

6) To modify the entity "rfree1": Click the ? at the bottom of the form and choose "rfree1". Use the vertical slider bar in the Command Form and scroll down until you see the FREE SURFACE DEPTH. Click FREE SURFACE DEPTH and input a value of -1 for the keyword DEPTH. *The value -1 (no spine generation) is used here because the maximum depth has already been specified for "lfree".* Click SPINE GENERATION and choose the MAPPED option. The updated form is shown below.



7) Move the cursor into the grey area, right click and choose REPLACE(REPEAT). Confirm the change was made correctly by checking the FIDAP Command History Window.

8) To modify the entity "rfree2": Click the ? at the bottom of the form and choose "rfree2". Use the vertical slider bar in the Command Form and scroll down until you see the FREE SURFACE DEPTH. Click FREE SURFACE DEPTH and input a value of 0 for the keyword DEPTH. Click SPINE GENERATION and choose the MAPPED option. Click CONTACT ANGLE 1 and retain the default value zero for the keyword ANG1. The updated form is shown below.

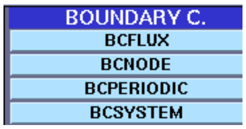
9) Move the cursor into the grey area, right click and choose REPLACE(REPEAT). Confirm the change was made correctly by checking the FIDAP Command History Window.

10) *In Step 13, the SLIPCOEFFICIENT was defined with the SET = "slip". The proper association with the corresponding entity must be performed. Recall that the SET on the SLIPCOEFFICIENT command must match the PROPERTY on the ENTITY command.* To modify the entity "slip": Click the ? at the bottom of the form and choose "slip". Click PROPERTY TYPE and click the ? to the right of "fluid" to select slip. The updated form is shown below.

11) Click REPLACE at the bottom of the form. Confirm the change was made correctly by checking the FIDAP Command History Window.

STEP 15: Define the Boundary Conditions

1) Click Boundary C. on the FIPREP Command Toolbar. The BOUNDARY C. Toolbar is shown below.



2) Click BCNODE on the BOUNDARY C. Toolbar. The BCNODE Command Form is shown below.

Table 3-6 describes the boundary conditions to be applied for the velocity degrees of freedom. Recall that the specific node numbers (needed for boundary conditions #6-#9) were determined in the FI-BC module in Step 3. It is important that boundary conditions #6-#9 are applied after boundary condition #1. The reason is that the tangential velocity at the dynamic contact point and the next closest nodes be free so that the Navier slip boundary condition will be applied at those nodes. If the order is reversed then the boundary condition #1 will overwrite boundary conditions #6-#9 and it is likely that convergence will not be achieved.

Table 3-6 Boundary Conditions for Velocity Components

B.C. #	REGION SELECTION	D.O.F.	VALUE
1	ENTITY = "slip"	UT	200
2	ENTITY = "slip"	UN	ZERO
3	ENTITY = "inlet"	UX	ZERO
4	ENTITY = "inlet"	UY	-200
5	ENTITY = "outlet"	UY	ZERO
6	NODE = 14	UX	FREE
7	NODE = 156	UX	FREE
8	NODE = 136	UX	FREE
9	NODE = 155	UX	FREE

Although the node numbers can be determined in FI-BC (or GAMBIT or FIPOST), this approach may become awkward if the geometry needs to be remeshed and the node numbers change. Note that the number of nodes to which the Navier slip condition is applied is generally 3-5. This method has the flexibility that the user can easily add (or delete) more boundary conditions of the type #9 after determining the correct node numbers if convergence is a problem.

A different approach would be to split the edge "slip" into two edges: "slip-1" and "slip-2". This is done in GAMBIT. The Navier slip condition would be applied to the entity containing the dynamic contact point: "slip-1". The boundary condition UT=200 would be applied to "slip-2". If convergence is a problem, the user would need to go back to GAMBIT to change the length of the entities.

3) For Boundary Condition #1: Change the DEGREE OF FREEDOM from UX to UT. (For SLIP type entities, UT is an alias for UX). The convention used in FIDAP is that the tangential direction is counter clockwise from the normal direction. (See the FIPREP manual, Chapter 7.4, (3) for more details.) Since the normal is in the - y direction, the tangential direction is in the + x direction. Change the REGION SELECTION keyword from NODES to ENTITY. Click on the ? to the right of ENTITY and choose slip. Input a value of 200 for the keyword CONSTANT. The updated BCNODE Command form is shown below.

4) Right click in the grey area in the BCNODE Command form and select ADD(REPEAT). Confirm the change was made correctly by checking the FIDAP Command History Window.

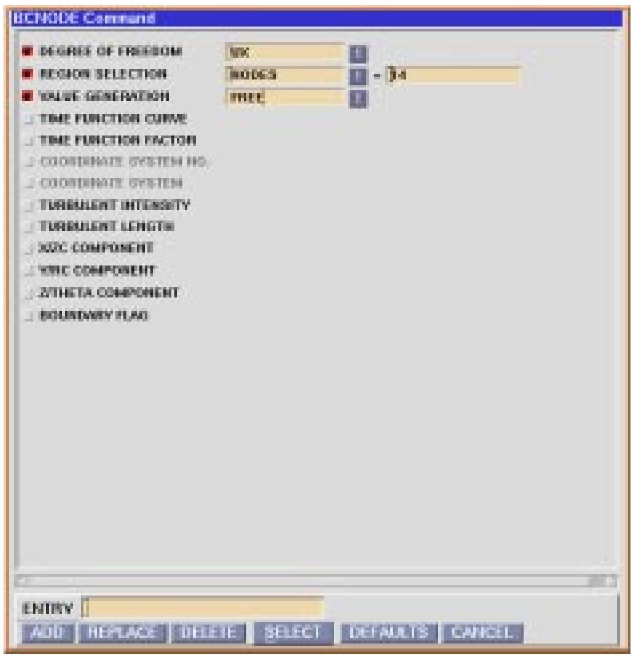
5) For Boundary Condition #2: Change the DEGREE OF FREEDOM keyword from UT to UN. (For SLIP entities, UN is an alias for UY). Change the value of the VALUE GENERATION keyword from CONSTANT to ZERO. The updated BCNODE Command form is shown below.

6) Right click in the grey area in the BCNODE Command form and select ADD(REPEAT). Confirm the change was made correctly by checking the FIDAP Command History Window.

7) Continue to apply boundary conditions #3, #4, #5 described in Table 3-6.

8) Right click in the grey area in the BCNODE Command form and select ADD(REPEAT). Confirm the change was made correctly by checking the FIDAP Command History Window.

9) For Boundary Condition #6: Change the DEGREE OF FREEDOM to UX. Change the REGION SELECTION to NODES. Input a value of 14 for keyword NODES. Change the VALUE GENERATION TO FREE. The updated BCNODE Command form is shown below.



10) Right click in the grey area in the BCNODE Command form and select ADD(REPEAT). Confirm the change was made correctly by checking the FIDAP Command History Window. Repeat this procedure for boundary conditions #7, #8 and #9 described in Table 3-6.

11) *FIDAP does not compute a physically correct normal direction at the corner nodes in the model. For further details, please refer to the FIPREP manual, Chapter 7.4, (2) for an explanation on how to apply physically correct normals using the COORDINATE degree of freedom and specifying the node number. Table 3-7 shows the Boundary Conditions for the COORDINATE DEGREE OF FREEDOM.*

Table 3-7: Boundary Conditions for
DEGREE OF FREEDOM = COORDINATE

B.C. #	REGION SELECTION	D. O. F.
10	NODE = 1	COORDINATE
11	NODE = 2	COORDINATE
12	NODE = 14	COORDINATE
13	NODE = 27	COORDINATE
14	NODE = 28	COORDINATE

For Boundary Condition #10: Change the DEGREE OF FREEDOM to COORDINATE. Input a value of 1 for the keyword NODES. Click off VALUE GENERATION. The updated form is shown below.

12) Right click in the grey area in the BCNODE Command form and select ADD(REPEAT). Confirm the change was made correctly by checking the FIDAP Command History Window. Repeat this procedure for boundary conditions #11, #12, #13 and #14 described in Table 3-7.

13) *The final two boundary conditions are needed to fix the free surface position at nodes 1 and 2. Table 3-8 shows the Boundary conditions needed for the SURFACE DEGREE OF FREEDOM. By specifying a ZERO Value for the SURFACE D.O.F. means that the node is fixed.*

Table 3-8: Boundary Conditions for DEGREE OF FREEDOM = SURFACE

B.C. #	REGION SELECTION	D. O. F.	VALUE
15	NODE = 1	SURFACE	ZERO
16	NODE = 2	SURFACE	ZERO

Boundary Condition #15: Change the DEGREE OF FREEDOM to SURFACE. Input a value of 1 for the NODES keyword. Click VALUE GENERATION and choose the keyword ZERO. The updated form is shown below.

14) Right click in the grey area in the BCNODE Command form and select ADD(REPEAT). Confirm the change was made correctly by checking the FIDAP Command History Window. Repeat for boundary condition #16 described in Table 3.8.

15) *The value of the contact angle at node 14 is generally determined from experiment.* Change the DEGREE OF FREEDOM to CONTACTANGLE. Input a value of 14 for the keyword NODES. Change the VALUE GENERATION keyword from ZERO to CONSTANT. Input a value of -20 for the keyword CONSTANT. Input a value of 1 for the keyword X (after the option X/ZC COMPONENT) to force the movement to be in the horizontal direction. The updated form is shown below.

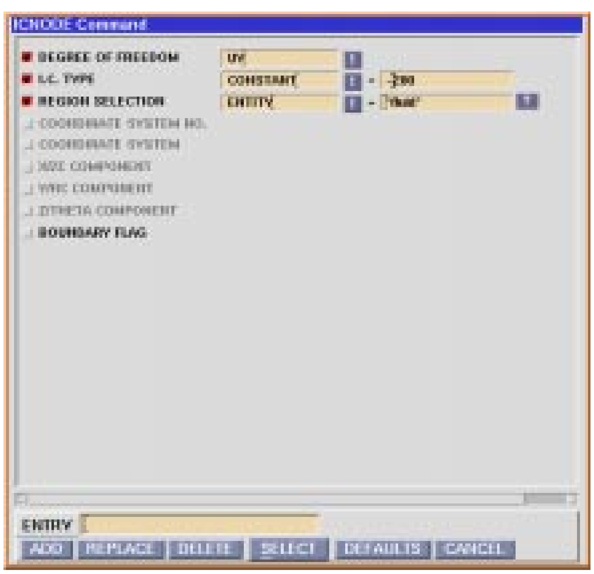
16) Click ADD at the bottom of the form. Confirm the change was made correctly by checking the FIDAP Command History Window.

STEP 16: Apply the Initial Conditions

The default initial condition is zero for all degrees of freedom. Typically a nonzero initial condition helps to prevent divergence or improve convergence.

1) Click Initial C. on the FIPREP Menu Toolbar. The ICNODE command Form is shown below.

2) Change the DEGREE OF FREEDOM keyword to UY. Change the I.C. TYPE keyword from ZERO to CONSTANT and input a value of -200. (You could have also specified UX = 200). Change the REGION SELECTION keyword from NODES to ENTITY and select fluid. The updated ICNODE Command Form is shown below.

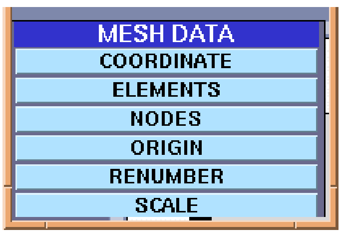


3) Click ADD at the bottom of the ICNODE Command Form. Confirm the change was made correctly by checking the FIDAP Command History Window.

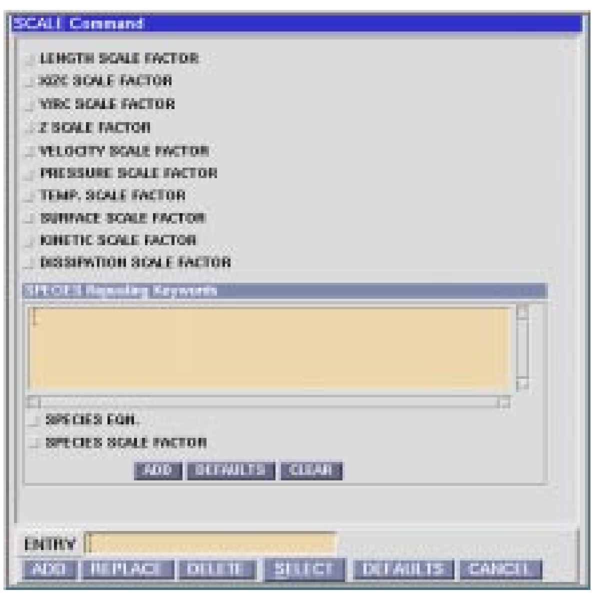
STEP 17: Scale the Geometry

The actual curtain thickness is .06 cm. To make the geometry easier to create in GAMBIT, the curtain thickness was created to be 1. Therefore, the entire geometry needs to be scaled by a factor of .06, which can be easily done in FIPREP. A scale command can be used to multiply all the dimensions in GAMBIT by any factor.

1) Click Mesh Data on the FIPREP Menu Toolbar. The MESH DATA Command Form is shown below.



2) Click SCALE on the MESH DATA form. The SCALE Command Form is shown below.



3) Click LENGTH SCALE FACTOR and enter a value of .06 for the keyword VALUE. The updated form is shown below.

4) Click ADD at the bottom of the form. Confirm the change was made correctly by checking the FIDAP Command History Window.

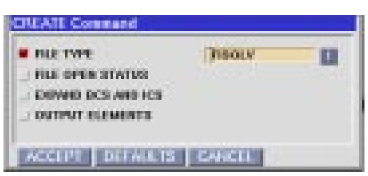
STEP 18: End out of the FIPREP Module

Click END on the FIDAP Command Toolbar.

STEP 19: Create the FISOLV Database

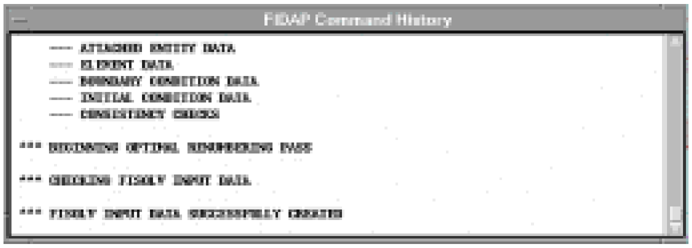
1) Click CREATE on the Main Module Toolbar. The Create Toolbar is shown below.

2) Click Create on the Create Toolbar. The CREATE Command Form is shown below:



The default FILE TYPE is FISOLV.

3) Click ACCEPT at the bottom of the CREATE Command Form. The FIDAP Command History Window will show the following message:



At the beginning of this tutorial, the .FIPREP file was read and a model database was created. This step updates that model database to also include the problem specifications: problem type, boundary conditions, physical properties, …. Before attempting to run the simulation, you need to check that the following message is reported in the FIDAP Command History Window:

```
***FISOLV INPUT DATA SUCCESSFULLY CREATED
```

If you get error messages, you can NOT proceed to STEP 20: Run the Simulation. To find the source of error you can go back into FIPREP and check the database or you can check the .FIINP file that you create next.

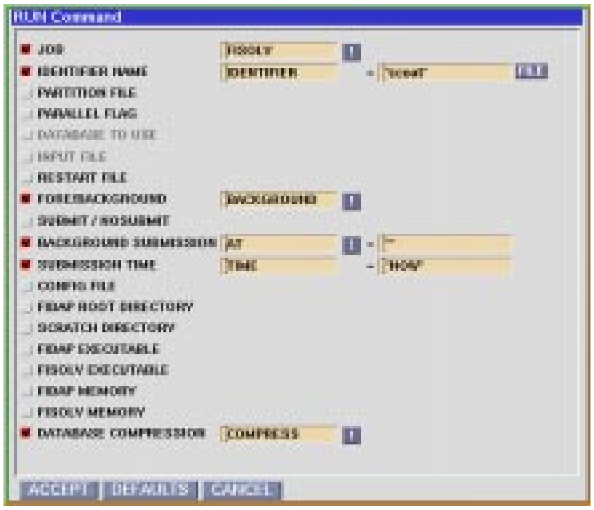
4) Click Create on the Create Toolbar. Change the FILE TYPE from FISOLV to FIPREP. Click FILE OPEN STATUS and select the keyword DELETE (to overwrite any preexisting .FIINP file). The updated CREATE Command Form is shown below.

5) Click ACCEPT at the bottom of the CREATE Command Form.

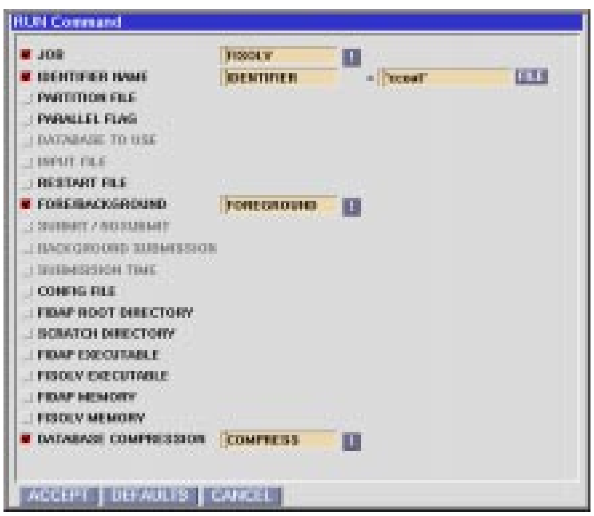
The file created is named ccoat.FIINP (see Appendix B). This file contains a clean set of commands and is very useful for diagnosing problems should convergence problems or inaccurate solutions arise. If the database was not created in the last step, this file should be checked for errors.

STEP 20: Run the Simulation

1) Click RUN on the Main Module Toolbar and select FISOLV. The RUN Command Form is shown below.



2) Change the RUN keyword from BACKGROUND to FOREGROUND. The updated RUN Command Form is shown below.



If you run the simulation in the background, the simulation will continue to run, even if you exit the FIDAP GUI. If you run the simulation in the foreground, you may display convergence data during the run (via the MONITOR option).

3) Click ACCEPT at the bottom of the RUN Command Form. The FOREGROUNDTASK IS RUNNING Form is shown below.

STEP 21: Monitor the Convergence

1) Click SOLUTION MONITOR at the bottom of the FOREGROUND TASK IS RUNNING Form. The SOLUTION MONITOR Form is shown below.

2) The FIDAP Command History Window will display the first message:

```
*** RUNNING FISOLV IN THE FOREGROUND WITH IDENTIFIER “ccoat”
```

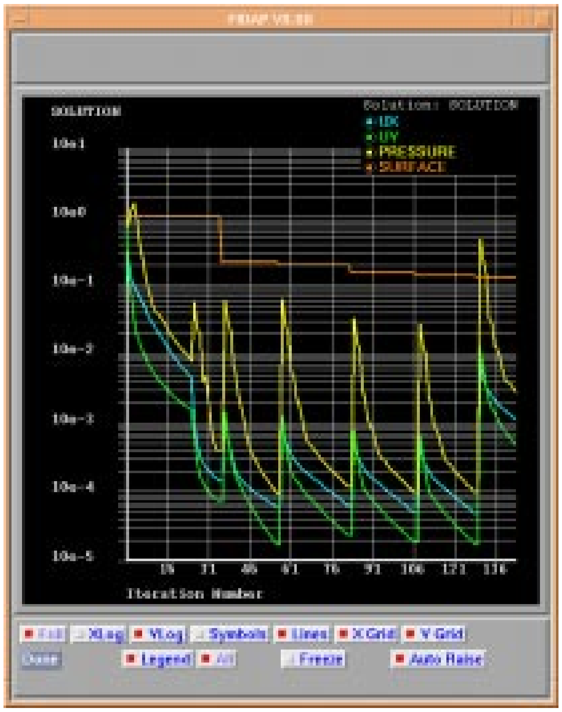
Wait for the following message to appear before proceeding to the next step.

```
Launching the fisolv job with the command line:……
```

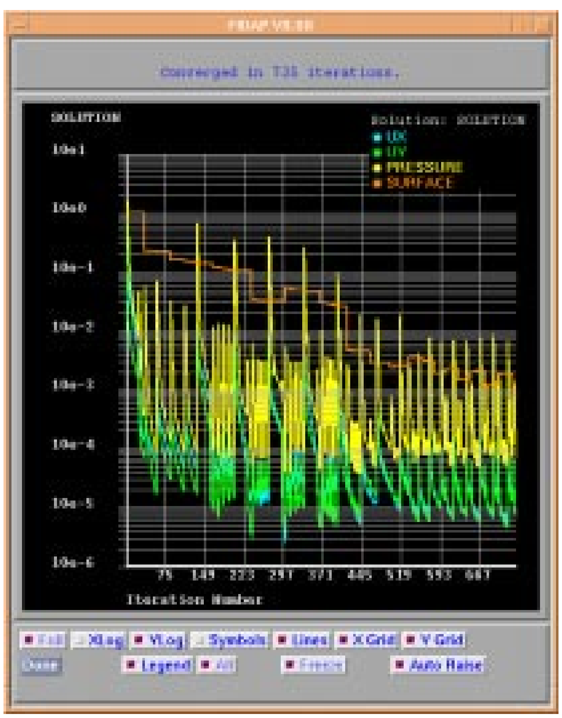
If you click START MONITOR too soon, you will get the error message in the FIDAP Command History Window:

```
Solution has not yet begun reporting convergence information.
```

3) Click START MONITOR at the bottom of the SOLUTION MONITOR Form. The Convergence Form is shown below. *A discussion of the convergence plot will be given in STEP 23.*



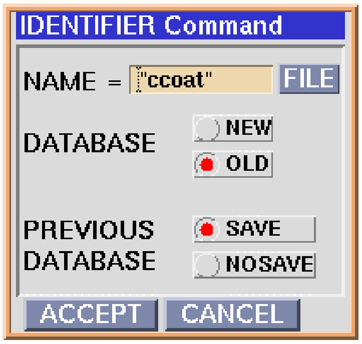
4) When the solution reaches convergence (735 iterations), click DONE at the bottom of the Convergence Form.



STEP 22: Reconnect the GUI to the database

FIDAP (where postprocessing is done) and FISOLV (where the solution is computed) are separate processes, so after running the simulation it is necessary to reconnect the GUI to the FIDAP database to postprocess the results.

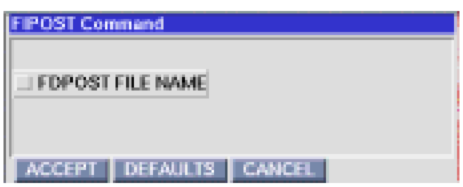
1) Click IDENT on the Main Module Toolbar. The IDENTIFIER Command Form is shown below.



2) Click ACCEPT at the bottom of the IDENTIFIER Command Form.

STEP 23: Postprocess the Results

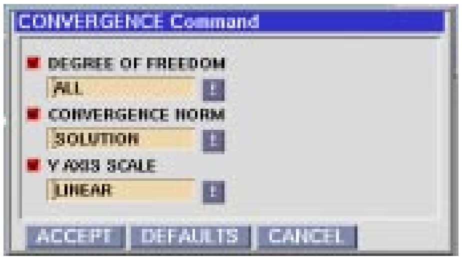
1) Click FIPOST on the Main Module Toolbar. The FIPOST Command Form is shown below.



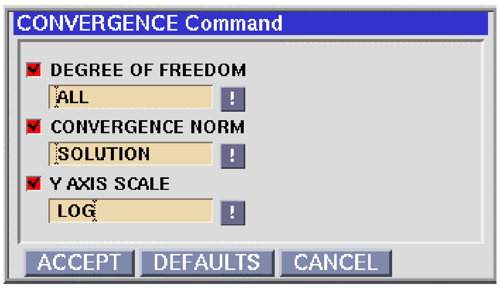
If you do not specify any FDPOST FILE NAME, the default file ccoat.FDPOST will be used. If you want to specify a file other than the default file, you will need to click the box to the left of FDPOST FILE NAME in the FIPOST Command Form and enter the file name of the results database.

2) Click ACCEPT at the bottom of the FIPOST Command Form. The FIPOST Menu Toolbar is shown below.

3) Check the Convergence. Click Convergence on the FIPOST Menu Toolbar. The CONVERGENCE Command Form is shown below.



Change the Y AXIS SCALE keyword from LINEAR to LOG. The updated CONVERGENCE Command is shown below.



Click ACCEPT at the bottom of the form. The Convergence Plot is shown below. The stairstep nature is typical of free surface convergence histories using the segregated solver in which the velocity field is solved until convergence, and then the free surface is updated, after which the velocity field must be solved again with the new free surface. This sequence is continued until both velocity and free surfaces are converged.

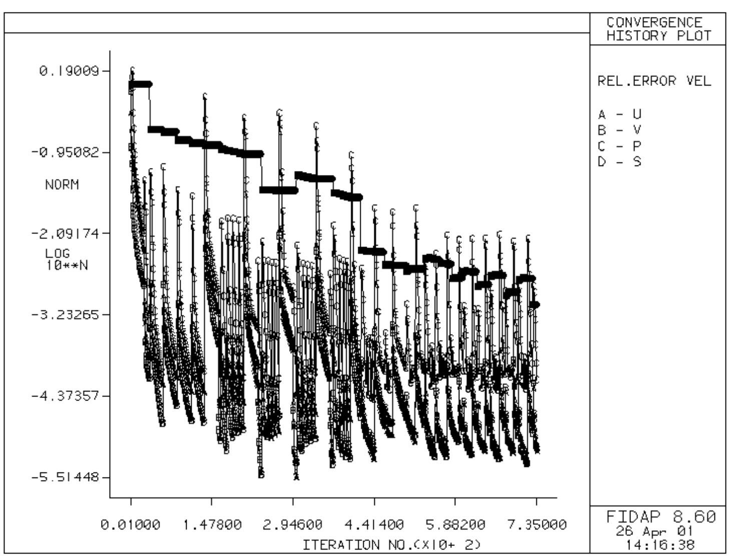


Figure 3-3: Convergence History

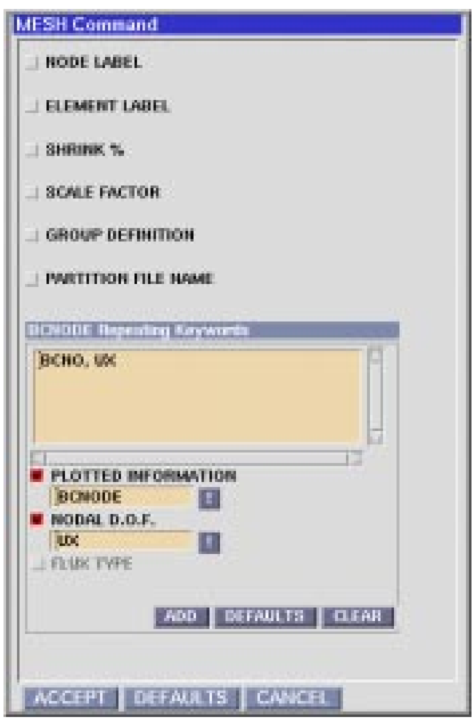
4) To specify a title for the plots, click on All in the FIPREP toolbar and select TITLE.

Click ACCEPT on the TITLE Command Form. The DATA NEEDED button will appear and you will be prompted to enter data. Press the F2 key and type in the Command Prompt Window:

Curtain Coating: Re = 2.85, Ca = 35.13

Press the Enter Key.

5) *Even though the solution has converged, it is a good idea to verify that the boundary conditions have been correctly applied. This can be done in the FIPOST module, however it is only a qualitative check. This step could have also been done after Step 19: creation of the DATABASE and before Step 20: run the simulation. It is also a good trouble shooting strategy if convergence is a problem.* Click Mesh on the FIPREP module. The MESH Command Form will appear. Click NODAL D.O.F. and change the keyword from VELOCITY to UX. Click ADD in the MESH Command form. The updated MESH Command Form is shown below.



Click ACCEPT in the MESH Command form. Slide the PAN/ZOOM bar (located in the Graphics Control Window) to the left slightly to better view the entire model. *The boundary conditions will be designated by arrows pointing to the right or upwards (indicating a positive boundary condition), arrows pointing to the left or downwards (indicating a negative boundary condition) or zeros (indicating a zero boundary condition). Arrows will be colored from red (largest magnitude) to blue (smallest magnitude). In the figure below, you will see red arrows pointing to the right at all of the nodes on the entity "slip", except for the first four nodes (indicating a positive ux boundary condition). There are zeros on all of the nodes at the "inlet" (indicating a zero UX boundary condition).*

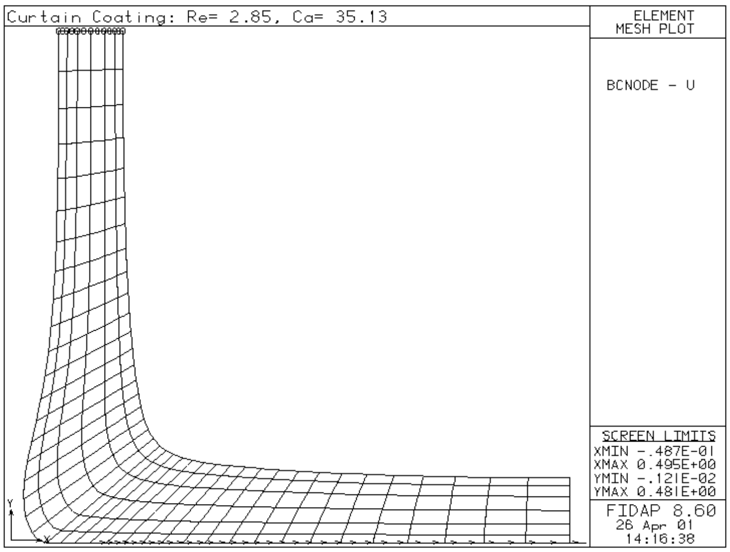


Figure 3-4: Mesh Plot and UX Boundary Condition.

Performing a similar check on the UY boundary conditions gives the plot shown below. There are red arrows pointing to the left at the "inlet" (indicating a large negative boundary condition for UY). There are zeros on all nodes on the "outlet" and "slip" (indicating a zero UY boundary condition).

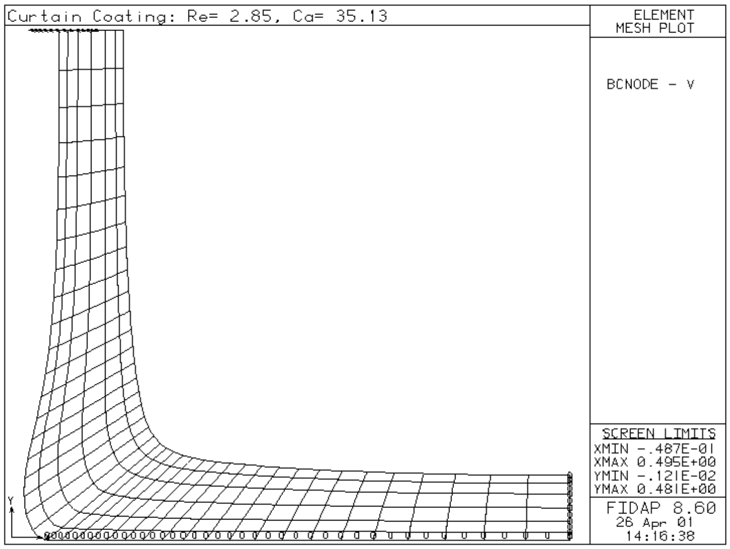


Figure 3-5: Mesh Plot and UY Boundary Condition.

6) Plot the free surface movement. Click FREE on the All Menu and click ACCEPT. The original shape is white. The final free surface shape is shown in green. The left free surface bulges to the left creating a small heel.

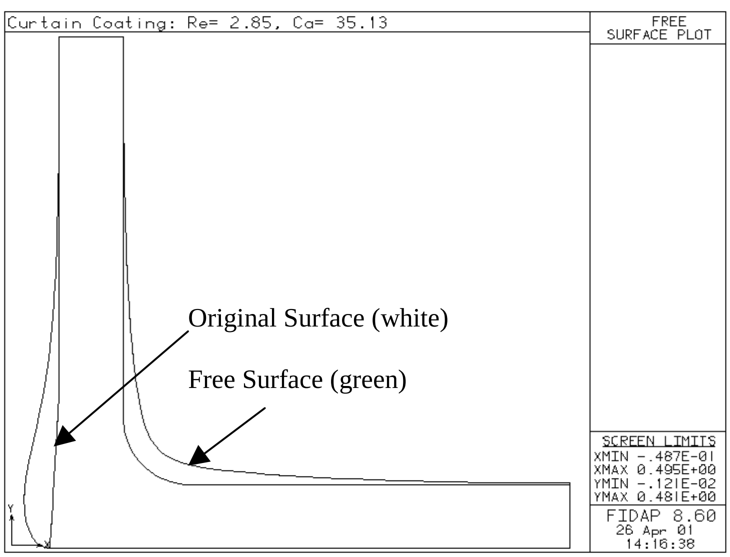


Figure 3-6: Movement of Free Surface

7) Plot the streamline contours. The Plot shows smooth contours in the entire region. There are no recirculation zones (closed contours) in the heel region.

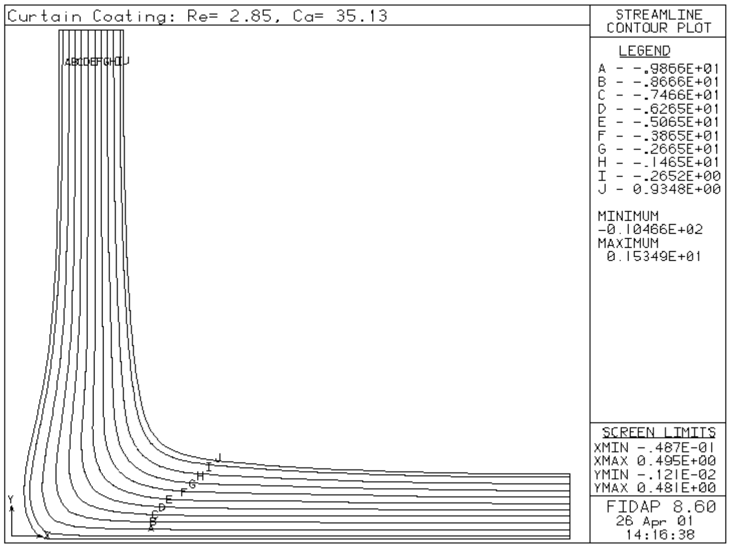


Figure 3-7: Streamline Contours

8) Create a Vector Plot: Click Vector on the FIPOST Menu Toolbar. Click ACCEPT on the bottom of the VECTOR Command Form. The Vector plot is shown below. You may need to use the PAN/ZOOM slider bar in the Graphics Control Window to scale the plot to fit.

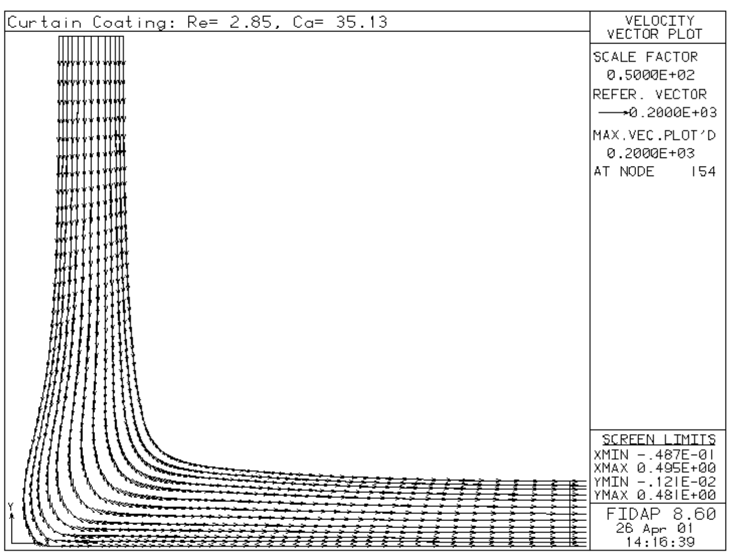


Figure 3-8: Vector Field

9) Plot the pressure contours. Note the spike in the pressure in the heel region. This is due to the velocity discontinuity at the dynamic contact point.

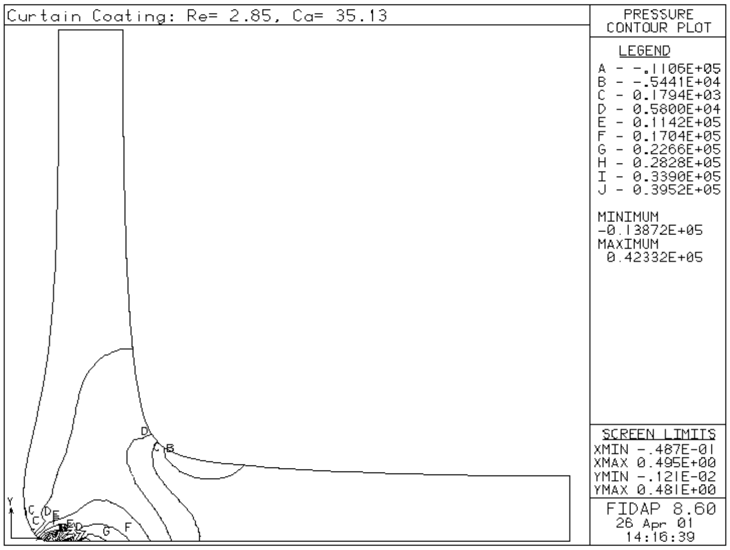


Figure 3-9: Pressure Contours

STEP 24: Exit the FIPOST module

Click END on the FIDAP Command Toolbar to end out of the FIPOST module and return to the FIDAP root level.

STEP 25: Exit FIDAP

Click END on the FIDAP Command Toolbar to end out of FIDAP. Click YES at the EXIT PROMPT Form.

SUMMARY:

In this tutorial the ratio of the speeds of the falling curtain and the moving substrate are equal. When the Reynolds number is low, FIDAP predicts a small heel formation. This is consistent with previously published results (1). The user can easily change the values: width and speed of the falling curtain, speed of the moving substrate, density, viscosity, and surface tension of the fluid for a new set of conditions. The recommended approach is to edit the file ccoat.FIINP (which was created in Step 19, #4) by uncommenting out some of the commands, inserting the necessary parameters and adding some commands. For more details on this procedure, please refer to Lecture 5 of the FIDAP Introductory Lecture Note. The basic steps are:

1. Copy the file ccoat.FIINP to the file ccoat_p.FDREAD and make the following changes to the file ccoat_p.FDREAD.

2. Uncomment out the 3 lines:
```
FICONV(NEUTRAL,NORESULTS,INPUT)
INPUT(FILE="ccoat.FDNEUT")
END
```

3. Insert the parameterized values. An example of just one of the values is given by:
```
$vel_c = 200
```

4. REPLACE the necessary FIPREP commands with the parameterized values.
   Change from
```
BCNO(UY, ENTI("inlet", CONS = -200.)
```
   to
```
BCNO(UY, ENTI="inlet", CONS = -$vel_c)
```

5. After the command PARAMETER (LIST), add the RUN Command:
```
RUN(FISOLV,FORE)
```

6. Insert the POSTPROCESSING commands:
```
IDENT(NAME = "ccoat_p")
FIPOST
DEVI(XWIN)
CONVERGENCE (ALL,LOG)
MESH(BCNO,UX)
MESH(BCNO(UY)
FREE
CONT (STREAMLINE)
VECTOR
CONTOUR (PRES)
END
```

An example input file (named ccoat_p.FDREAD) is provided in Appendix C.

REFERENCES:

1. Kistler, Stephan F. and Schweizer, Peter M. (1997), Liquid Film Coating: Scientific principles and their technological implications, Chapman and Hall, London.

APPENDIX A: ccoat.FIPREP

```
/ ****************************************************************
/ Disclaimer: This file was written by GAMBIT and contains
/ all the continuum and boundary entities and coordinate systems
/ defined in GAMBIT. Additionally, some frequently used FIPREP
/ commands are added. Modify/Add/Uncommment any necessary commands.
/ Refer to FIPREP documentation for complete listing of commands.
/ ****************************************************************
/
/          CONVERSION OF NEUTRAL FILE TO FIDAP Database
/
FICONV( NEUTRAL )
INPUT( FILE="ccoat.FDNEUT" )
OUTPUT( DELETE )
END
/
TITLE
ccoat
/
FIPREP
/
/          PROBLEM SETUP
/
PROBLEM (2-D, LAMINAR, NONLINEAR, ISOTHERMAL)
EXECUTION( NEWJOB )
PRINTOUT( NONE )
DATAPRINT( CONTROL )
/
/          CONTINUUM ENTITIES
/
ENTITY ( NAME = "fluid", FLUID, PROPERTY = "fluid" )
/
/          BOUNDARY ENTITIES
/
ENTITY ( NAME = "inlet", PLOT )
ENTITY ( NAME = "outlet", PLOT )
ENTITY ( NAME = "slip", SLIP  )
ENTITY ( NAME = "lfree", SURFACE )
ENTITY ( NAME = "rfree1", SURFACE )
ENTITY ( NAME = "rfree2", SURFACE )
/
/          LOCAL COORDINATE SYSTEMS DEFINED
/
/COORDINATE ( SYSTEM = 2, MATRIX,CARTESIAN )
/0.000000 0.000000 0.000000 1.000000 0.000000 0.000000 0.000000
1.000000 0.000000 0.000000 0.000000 1.000000
/
```

```
/          SOLUTION PARAMETERS
/
/SOLUTION( S.S. = 10, VELCONV = .01, RESCONV = .01, ACCF = .0 )
/PRESSURE( PENALTY = 1.E-6, DISCONTINUOUS )
/OPTIONS( UPWINDING, , , )
/SCALE( VALUE = 1 )
/TIMEINTEGRATION( BACKWARD,NSTEPS = ,DT = 0,,, )
/POSTPROCESS( NBLOCKS =  )
/
/          MATERIAL PROPERTIES
/
/ Partial list of Material Properties data
/
/The following lines were uncommented.  This step will not be necessary
/in GAMBIT versions 2.0 and later.
DENSITY( SET = "fluid", CONSTANT = 1 )
VISCOSITY( SET = "fluid", CONSTANT = 1 )
/CONDUCTIVITY( SET = "fluid", CONSTANT = 1 )
/SPECIFICHEAT( SET = "fluid", CONSTANT = 1 )
/
/          INITIAL AND BOUNDARY CONDITIONS
/
/ICNODE( , CONSTANT = 0, ALL )
/
/BCNODE( , CONSTANT = 0, ENTITY = "inlet" )
/BCNODE( , CONSTANT = 0, ENTITY = "outlet" )
/BCNODE( , CONSTANT = 0, ENTITY = "slip" )
/BCNODE( , CONSTANT = 0, ENTITY = "lfree" )
/BCNODE( , CONSTANT = 0, ENTITY = "rfree1" )
/BCNODE( , CONSTANT = 0, ENTITY = "rfree2" )
/
END
/
CREATE( FIPREP,DELETE )
PARAMETER( LIST )
CREATE( FISOLV )
/RUN( FISOLV, FOREGROUND )
```

APPENDIX B: ccoat.FIINP

```
/
/  FICONV(NEUTRAL,NORESULTS,INPUT)
/  INPUT(FILE= "ccoat.FDNEUT")
/  END
/
TITLE
/
FIPREP
 PROB (2-D, STEA, INCO, LAMI, NONL, NEWT, MOME, ISOT, FREE, SING)
 PRES (MIXE, DISC)
 EXEC (NEWJ)
 SOLU (SEGR = 100, VELC = 0.100000000000E-03, SURF = 0.100000000000E-02,
       PREC = 21, KINE = 500)
 DATA (NONE)
 RELA (HYBR)
   0.3000000000E+00,  0.3000000000E+00,  0.0000000000E+00,  0.5000000000E+00,
   0.0000000000E+00,  0.9750000000E+00,  0.0000000000E+00,  0.0000000000E+00,
   0.0000000000E+00,  0.0000000000E+00,  0.0000000000E+00,  0.0000000000E+00,
   0.0000000000E+00,  0.0000000000E+00,  0.0000000000E+00,  0.0000000000E+00,
   0.0000000000E+00,  0.0000000000E+00,  0.0000000000E+00,  0.0000000000E+00,
   0.0000000000E+00,  0.0000000000E+00,  0.0000000000E+00,  0.0000000000E+00,
   0.0000000000E+00,  0.0000000000E+00,  0.0000000000E+00,  0.0000000000E+00
 PRIN (NONE)
 SCAL (VALU = 0.600000000000E-01)

 ENTI (NAME = "fluid", FLUI, PROPERTY ="fluid")
 ENTI (NAME = "inlet", PLOT)
 ENTI (NAME = "outlet", PLOT)
 ENTI (NAME = "slip", SLIP, PROPERTY = "slip")
 ENTI (NAME = "lfree", SURF, DEPT = 0, X = 1.0, MAPP)
 ENTI (NAME = "rfree1", SURF, DEPT = -1, MAPP, NORM)
 ENTI (NAME = "rfree2", SURF, DEPT = 0, MAPP, NORM, ANG1 = 0.0)

 DENS (SET = "fluid", CONS = 1.25)
 VISC (SET = "fluid", CONS = 5.27)
 SURF (SET = "fluid", CONS = 30.0)
 SLIP (SET = "slip", CONS = 0.100000000000E-01, UT = 200.0)
```

```
 BCNO (UT, ENTI = "slip", CONS = 200.0)
 BCNO (UN, ENTI = "slip", ZERO)
 BCNO (UX, ENTI = "inlet", ZERO)
 BCNO (UY, ENTI = "inlet", CONS = -200.0)
 BCNO (UY, ENTI = "outlet", ZERO)

 BCNO (CONT, CONS = -20.0, NODE = 14, X = 1.0, Y = 0.0)

 BCNO (UX, NODE = 14, FREE)
 BCNO (UX, NODE = 156, FREE)
 BCNO (UX, NODE = 136, FREE)
 BCNO (UX, NODE = 155, FREE)

 BCNO (COOR, NODE = 1)
 BCNO (COOR, NODE = 2)
 BCNO (COOR, NODE = 14)
 BCNO (COOR, NODE = 27)
 BCNO (COOR, NODE = 28)
 BCNO (SURF, NODE = 1, ZERO)
 BCNO (SURF, NODE = 2, ZERO)

 ICNO (UY, CONS = -200.0, ENTI = "fluid")
END
/  *** of FIPREP Commands
CREATE(FIPREP,DELE)
CREATE(FISOLV)
PARAMETER(LIST)
```

APPENDIX C: ccoat_p.FDREAD

```
/ccoat_p.FDREAD
/This is a parameterized file for a curtain coating problem.
/This file both runs the simulation and postprocesses the results.
/To run this file type the command:
/fidap -id ccoat_p -in ccoat_p.FDREAD -new
/Note: The file ccoat.FDNEUT must also be in your working directory.
/

FICONV(NEUTRAL,NORESULTS,INPUT)
INPUT(FILE= "ccoat.FDNEUT")
END
/
/units are CGS
/
/scale geometry in GAMBIT so that characteristic length at outlet
/  is .06 cm
/length units are cm
$length = .06

/scale factor
$scfac = $length

/density units are g/m3
$density = 1.25

/viscosity units are g/(cm-s)
$viscosity = 5.27

/surface tension units are g/s2
$surftens = 30.0

/velocity units of falling curtain and moving substrate are cm/s
$vel_c = 200
$vel_s = 200

/slip coefficient
$slipcoef = .01

/contact angle units are degrees
$contangle = 20

/Reynolds number and Capillary number at "outlet"
$Re = $density * $vel_s * $length / $viscosity
$Ca = $viscosity * $vel_s / $surftens

/Relaxation values for velocity, pressure and free surface d.o.f.
$vel_rel = .3
$pre_rel = .5
$sur_rel = .975
```

```
/title information
$fr = 100
$fc = 100
$Re1 = 100 * $Re
$Re2 = INT ($Re1)
$Re3 = ($Re2/$fr)

$Ca1 = 100 * $Ca
$Ca2 = INT ($Ca1)
$Ca3 = ($Ca2/$fc)

$head = "Curtain Coating:"
$rtitle = " Re = " + NTOS ($Re3)
$ctitle = " Ca = " + NTOS ($Ca3)

TITLE (String)
($head + $rtitle + $ctitle)

FIPREP
 PROB (2-D, STEA, INCO, LAMI, NONL, NEWT, MOME, ISOT, FREE, SING)
 PRES (MIXE, DISC)
 EXEC (NEWJ)
 SOLU (SEGR = 100, VELC = 0.1E-03, SURF = 0.1E-02, PREC = 21, KINE =
500)
 DATA (NONE)
 RELA (HYBR)
/ U        V       W  P        T S
 $vel_rel $vel_rel 0 $pre_rel  0 $sur_rel
 PRIN (NONE)
 SCAL (VALU = $scfac)

 ENTI (NAME = "fluid", FLUI, PROP = "fluid")
 ENTI (NAME = "inlet", PLOT)
 ENTI (NAME = "outlet", PLOT)
 ENTI (NAME = "slip", SLIP, PROP = "slip" )
 ENTI (NAME = "lfree",  SURF, DEPT = 0,  MAPP, X = 1.0)
 ENTI (NAME = "rfree1", SURF, DEPT = -1, MAPP, NORM)
 ENTI (NAME = "rfree2", SURF, DEPT = 0,  MAPP, NORM, ANG1 = 0.0)

 DENS (SET = "fluid", CONS = $density)
 VISC (SET = "fluid", CONS = $viscosity)
 SURF (SET = "fluid", CONS = $surftens)
 SLIP (SET = "slip",  CONS = $slipcoef, UT = $vel_s)
```

```
 BCNO (UT, ENTI = "slip",   CONS = $vel_s)
 BCNO (UN, ENTI = "slip",   ZERO)
 BCNO (UX, ENTI = "inlet",  ZERO)
 BCNO (UY, ENTI = "inlet",  CONS = -$vel_c)
 BCNO (UY, ENTI = "outlet", ZERO)
 BCNO (CONT, CONS = $contangle, NODE = 14, X = 1.0, Y = 0.0)
 BCNO (UX, NODE = 14, FREE)
 BCNO (UX, NODE = 156, FREE)
 BCNO (UX, NODE = 136, FREE)
 BCNO (UX, NODE = 155, FREE)
 BCNO (COOR, NODE = 1)
 BCNO (COOR, NODE = 2)
 BCNO (COOR, NODE = 14)
 BCNO (COOR, NODE = 27)
 BCNO (COOR, NODE = 28)
 BCNO (SURF, NODE = 1, ZERO)
 BCNO (SURF, NODE = 2, ZERO)

 ICNO (UY, CONS = -$vel_c, ENTI = "fluid")
END

CREATE(FIPREP,DELE)
CREATE(FISOLV)
PARAMETER(LIST)
RUN(FISOLV,FORE)

IDENT(NAME="ccoat_p")

FIPOST
DEVI(XWIN)
CONVERGENCE( ALL, LOG )
MESH( BCNO, UX )
MESH( BCNO, UY )
FREE
CONT(STREAMLINE)
VECTOR(  )
CONTOUR( PRES )
END
```