

date: February 29, 2000
Original September 9, 1998; Revised February 29, 2000 (PRS)

to: Distribution

from: P. R. Schunk and Matt Stay 9111, MS0826

subject: Roll coating templates and tutorial for GOMA and SEAMS (GT-003.2)

keywords: forward roll coating, deformable roll coating, fluid-structure interactions, film-split, volume constraints, automated continuation, rolling bank

inputrecords: Lamé MU, Lamé LAMBDA, Mesh Motion=LAGRANGIAN, Mesh Motion=ARBITRARY, Convective Lagrangian Velocity=ROTATIONAL, NO_SLIP, SOLID_FLUID, FLOW_PRESSURE, AC=VC, SURFTANG, SURFTANG_SCALAR, CAPILLARY

Introduction

This tutorial assumes the user has gone through the beginner's training tutorial on GOMA and SEAMS. Specifically, it assumes the user has a strong familiarity with GOMA itself and the tools **fastq**, **ex1ex2v2**, **aprepro** and **blot**. If not, please go through that tutorial first if you have it, or contact Duane Labreche (dalabre@sandia.gov) or Randy Schunk (prschun@sandia.gov) to get it.

We will introduce roll coating in three stages: (1) flooded nip with deformable rolls, (2) film split problem with a flooded upstream nip (on rigid rolls) both forward and reverse mode, and (3) finally a full-up forward roll coating problem with both upstream and downstream menisci. The name of the directory to which this portion of the tutorial refers is the **forward.9** directory. Contact us for the most recent copy of this, or look on the website. At this time we have not finished a template on full-up reverse roll coating.

The key to running all of these problems is to obtain a good initial guess and then knowing the details of the problem parameterization. This tutorial will try to cover all phases.

NOTE: THIS MEMO REPLACES AN EARLIER VERSION OF THE ROLL COATING TUTORIAL, WITH A NEW PROCEDURE FOR THE FULL FORWARD ROLL CASE. THE APPROPRIATE TEMPLATE FOR THAT CASE IS CALLED **forward.9**.

Floded Deformable Roll Model

Basically here we are immersing the rolls in liquid and predicting the interaction of the liquid and rollers, including the roller deformation. This is the simplest of all roll coating models. You should have a directory in “roll” is called “def_roll_nip”. We have updated this a bit from the original distribution so contact Duane Labreche (dalabre@sandia.gov) or Randy Schunk (prschun@sandia.gov) for an updated copy, if yours is older than 1 Oct. 1997.

In that directory you have the following files:

```
contin.both_soft  input.lower_soft  roll.exoII      upper_roll.mat
contin.dat        liquid.mat        roll.fas
contin.lower_soft lower_roll.mat    roll.gen
input.both_soft  out.exoII        roll_input
```

The important thing here is that there are 3 material files, **upper_roll.mat**, **lower_roll.mat** and **liquid.mat**. We will do a simulation with one effectively rigid roll, the upper one, and one soft roll, the lower one. We will then move the rolls together so we have effectively what is termed a “compression”, or a negative gap.

To get an initial solution here, edit first both the **upper_roll.mat** and **lower_roll.mat** and make sure that the **lame_mu** and **lame_lambda** cards are as follows:

```
Lame MU           = CONSTANT   1000000.
Lame LAMBDA       = CONSTANT    0.
```

Notice here that the solid constitutive equation is **INCOMP_PSTRAIN**, which is an incompressible plane strain model and thus requires only a shear modulus, as Poisson’s ratio is 0.5. The large number effectively makes the rollers rigid. We found that it is difficult to get convergence from a trivial (zero) initial guess to the deformable case owing to the nonlinearities of the problem. Now make sure that your initial guess card in “roll_input” is set to “zero”, viz.

```
Initial Guess      = zero
```

and run GOMA,

```
goma -a -i roll_input
```

Save the solution with “**cp soln.dat contin.dat**” and then edit **roll_input** and change the Initial guess card to “**read**”. Also edit the **upper_roll.mat** file and change the **Lame Mu** coefficient as follows:

```
Lame MU           = CONSTANT   100000.
```

Notice here we have lowered the modulus by one order of magnitude. Rerun GOMA:

```
goma -a -i roll_input
```

Continue these steps by lowering the **Lame Mu** coefficient of the upper roll in the following sequence:

from 100000 to 50000 and then from 50000 to 30000.

-3-February 29, 2000

Distribution

Use **blot** to look at the results. Notice the deformation the hydrodynamic forces of the liquid imparts on the upper roll. Now is worth explaining some of the key BCs that are needed to accomplish this problem. In the **roll_input** file you will notice that we are solving the moving mesh equations in the upper and lower roll materials and that the mesh motion of the materials is set to “**LAGRANGIAN**”, e.g.,

```
MAT = lower_roll 1
```

```
Coordinate System = CARTESIAN
Element Mapping = isoparametric
Mesh Motion = LAGRANGIAN
Number of bulk species = 0
```

```
Number of EQ          = 2
EQ = mesh1   Q2   D1   Q2  0.  0.  1.  1.   0.  0.
EQ = mesh2   Q2   D2   Q2  0.  0.  1.  1.   0.  0.
```

```
div ms adv bnd dif src porous
```

The liquid material, Block 3, on the other hand, has a mesh motion scheme set at “**ARBITRARY**”. The reason that solid materials are treated as computational Lagrangian materials is to be consistent with the solid momentum equation formulation. Moreover, there is a superimposed motion specified on the rolls in the material files. Notice in **lower_roll.mat** and **upper_roll.mat** there the following cards:

```
Convective Lagrangian Velocity = ROTATIONAL {-rollsp_t} {x11} {y11} 0
or
Convective Lagrangian Velocity = ROTATIONAL {-rollsp_b} {x1} {y1} 0
```

These cards specify that there is a advective term put on the solid momentum equations that will account for the solid body rotation of the roll, appropriately corrected for the deformation it undergoes. i.e. the term takes the user-specified velocity field and projects it into the deformation gradient tensor term to get the proper accounting of the kinematics and stresses. The model here on this card is “**ROTATIONAL**”, and the input required is the rotation rate (in Radians/sec, i.e., the **rollsp_t** APREPRO variable defined in the **roll.fas** file), and the center of rotation, which is either point 1 or 11, depending on which material file. Check out the geometry in **fastq** to see, i.e., **fastq -a roll.fas**.

Now, back in the **roll_input** file, the boundary conditions on the roll surfaces are specified as follows:

```
Boundary Condition Specifications
```

```
---
```

```
Number of BC          = -1
BC = PLANE SS 5 1. 0. 0. {-x5}
BC = PLANE SS 6 1. 0. 0. {x5}
```

```
$ Right now do nothing regarding flow
$ at inflow and outflow bndrys
```

```
BC = DX   NS  150  0.0
BC = DY   NS  150  0.0
BC = DX   NS   50  0.0
BC = DY   NS   50  0.0
```

Distribution

```

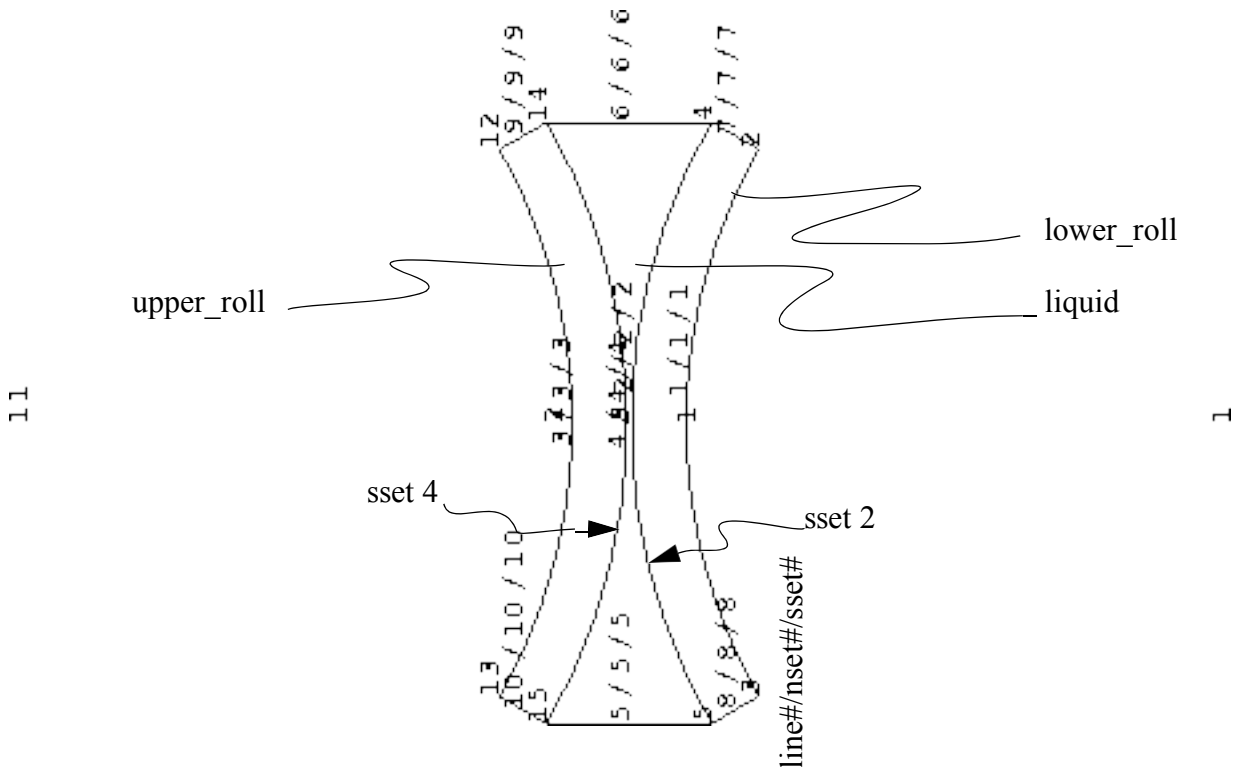
$
$ Roll inlets and outlets and cover/roll interface
$
BC = DX   NS 10  0.0
BC = DY   NS 10  0.0
BC = DX   NS  3  0.0
BC = DY   NS  3  0.0
BC = DX   NS  1  0.0 1.0
BC = DY   NS  1  {Gap - Gap_new} 1.0
BC = DX   NS  7  0.0 1.0
BC = DY   NS  7  {Gap - Gap_new} 1.0
BC = DX   NS  8  0.0 1.0
BC = DY   NS  8  {Gap - Gap_new} 1.0
BC = DX   NS  9  0.0
BC = DY   NS  9  0.0

BC = NO_SLIP  SS 2 1 3
BC = SOLID_FLUID SS 2 1 3

BC = SOLID_FLUID SS 4 2 3
BC = NO_SLIP  SS 4 2 3

#####
END OF BC
#####
    
```

These BCs correspond to the following geometry, where the node and side sets are labeled:



The noteworthy BCs are the **SOLID_FLUID** cards on side sets 2 and 4. There the stresses are balanced between the liquid and solid, resulting in the roll deformation. the integer data following these cards

are respectively `side_set_no.`, `solid_material_no.`, `liquid_material_no.` Notice that the lower roll is block/material number 1, the upper roll block/material number 2, and the liquid material/block number 3. The `NO_SLIP` cards simply set the liquid velocity to the solid velocity along that surface.

Now we will press the rolls together during operation. Obviously you cannot do this from the start as the mesh can not be generated on a negative gap. This is done differently for the case in which the rolls are being modeled as a deformable moving solid than when we are modeling them just as a rigid surface, see below. We can approach this in one of two ways (if you are sick of reading, go to the second approach here as it works MUCH better):

- 1 We will simply give the whole roll an initial displacement from the original stress free state. The easiest is to give the lower roll (or lower roll) a displacement independently of the rest of problem. In this case the upper roll has in it already some deformation, so we cannot move the upper roll here. This approach would obviously not apply to the case when BOTH rolls are deformable. THE-RULE-OF-THUMB, ALWAYS PARAMETERIZE YOUR PROBLEMS SO THE THAT RIGID ROLL (OR COATING DIE IN THE CASE OF SLOT COATING OR SOMETHING LIKE THAT) MOVES. There are two places which initializations can be made: in the GOMA input file and in the material files. Any initialization in the GOMA input file applies to *all* materials, so that is not the route we want to use. Lets look at the bottom of the `lower_roll.mat` file:

```
$$Initialize = MESH_DISPLACEMENT2 0 {Gap_new-Gap}
```

This card is commented out. If uncommented by removing the “\$\$” and setting `Gap_new` at the top of `roll.fas` to a slightly higher or lower value will displace the lower roll by that amount. I tried this and frankly it does not work too well as the changes have to be VERY small so as not to overtake the elements in the liquid phase too abruptly.

- 2 The better way to do this is evident from the displacement BC’s in the `roll_input` file. Note that on node sets 1, 7 and 8 there are displacement boundary conditions as follows:

```
BC = DX   NS   1   0.0 1.0
BC = DY   NS   1   {Gap - Gap_new} 1.0
BC = DX   NS   7   0.0 1.0
BC = DY   NS   7   {Gap - Gap_new} 1.0
BC = DX   NS   8   0.0 1.0
BC = DY   NS   8   {Gap - Gap_new} 1.0
```

The Y displacement of the lower roll will be affected by moving the rigid core/rubber interfaces, and the artificial ends of the roll. The second floating point number on these cards (after the displacement) is the Newton relaxation parameter. All Dirichlet conditions in GOMA can handle this factor. If present, GOMA includes these conditions in the matrix and iterates on them “softly”. If not present, they are “hard” set, in which case you can only take minuscule steps to avoid mesh distortion. In `BLOT` or in the little figure above you can see which surfaces these are. So try the following displacement in the `roll.fas` file (N.B. this file is included in the “`roll_input`” file and there is no need to regenerate the mesh):

```
$ gap (leading edge to substrate)   {Gap = 0.001} {Gap_new = 0.001}
```

to

```
$ gap (leading edge to substrate)   {Gap = 0.001} {Gap_new = 0.0004}
```

Then run GOMA (“`goma -a -i roll_input`”) and look at the results and make sure you remember to “`cp soln.dat contin.dat`”.

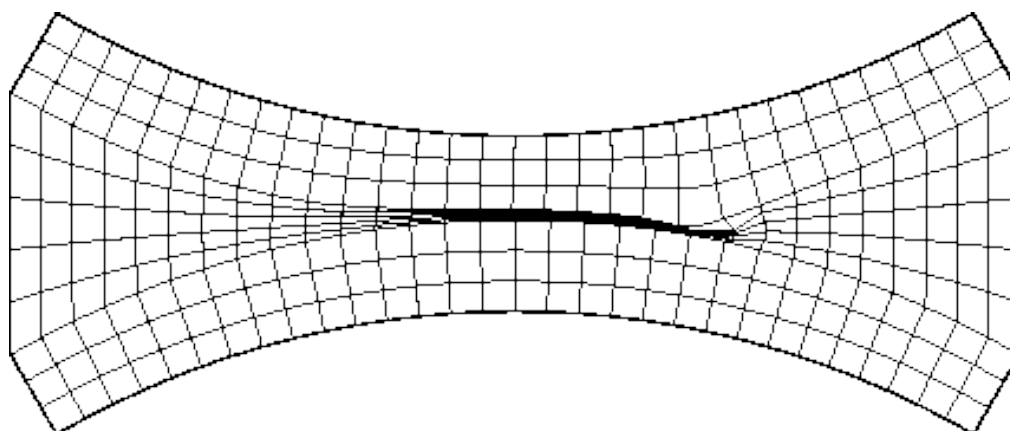
Now go another level:

```
$ gap (leading edge to substrate)    {Gap = 0.001} {Gap_new = 0.000}
```

Run GOMA. Copy the solution file to the continuation file (`cp soln.dat contin.dat`), increase the compression,

```
$ gap (leading edge to substrate)    {Gap = 0.001} {Gap_new = -0.0005}
```

and run the same sequence. I got down to about -0.0012 (i.e., a negative 1.2 mm gap) in small increments before the mesh got too distorted. To go much further you would have to add a user-defined `lame_mu` constant in the liquid phase which keeps the mesh stiff in the nip, so it doesn't get squeezed out. The figure below illustrates what you should see.



Film Split Roll Coating Model

In the “roll” subdirectory (viz. `$(INSTALL_DIR)/Distribution/example_problems/roll`) there is a `film_split_rigid` subdirectory. In there you'll find 2 key files: `roll_input` and `roll.fas`. First lets look at `roll.fas`.

The top of this file looks like this:

```
TITLE
Flooded-roll nip with film split

$ GEOMETRY AND OPERATING CONDITIONS (MKS)

$ Note here roll speed for the top roll are in m/s because
```

-7-February 29, 2000

Distribution

\$ of the type of bc that we use.

```

$ gap                {Gap = 0.001} {Gap_new = 0.001}
$ roll cover thickness (1'' = 0.0254) {d = 0.0254/2.}
$ bottom roll speed  {rollsp_b = 0.2}
$ top roll speed     {rollsp_t = 0.1}
$ initial film thickness for v2/v1=1 {h_t = .002}

$ Roll Radius        {R_I = 7.*0.0254} {R_I_new = 6.8*0.0254}
$ Outer Roll Radius  {R_O = R_I + d} {R_O_new = R_I_new+d}
$ Outer Roll Radius + film thickness {R_O_h = R_O + h_t} {R_O_h_new = R_O_new + h_t}
$ Outer Roll Radius + 2.5*film thickness {R_O_2h = R_O + 2.5*h_t}

$ Roll extent {theta1 = 60} {theta2 = 120}

$
$ MESH PARAMETERS
$
${no_elem_along_roll = 30}
${no_elem_across_roll_cover = 3}
${no_elem_outflow = 5}
${no_elm_along_roll_for_film_split = 20}
${no_elm_between_films = 10}
${no_elm_across_film = 3}

```

Note here that this part of the file simply describes the geometry of the rolls, the operating conditions, and sets the density of the finite element discretization. In other words, the whole problem is parameterized in this section. Noteworthy are the units, which are all in MKS and the **APREPRO** variables **Gap_new**, **R_I_new**, and the other ***_new** variables. These are the geometrical changes that can be made in the file during a run, i.e, after an initial solution has been obtained. It is important to understand here that GOMA solves for displacements of the nodes and not the actual coordinates as they evolve, and so coordinate DIFFERENCES must be tracked instead of absolute values. Hence, the boundary condition changes for Gap and roll radius alterations must have both the initial value and the desired value, the initial value being that at which the mesh was originally generated.

The “**roll extent**” line, and **no_elem*** lines can be changed to model a larger part of the roll surface with more or fewer elements. We suggest you simply play with those to get a good feel for what they do.

The rest of the file describes a geometry with a series of **fastq** commands. Note the extensive use of **APREPRO**. Noteworthy is the definition of **y11_new** on line 44. This variable basically is used to track the requested change in the centroid of upper roll, as [x11, y11] are the coordinates of the centroid of that roll, which changes for gap and roll radius changes. **y11_new** is used in **roll_input**, the GOMA input file. Also noteworthy are the following lines which define points 22, 30, 32, and 24:

```

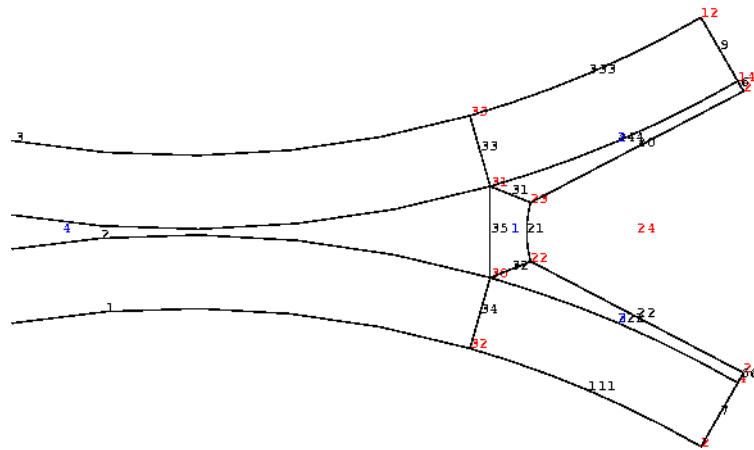
point 22 {x22=R_O_h*cosd(theta1+14)} {y22 = R_O_2h*sind(theta1+14)}

point 30 {x30= R_O*cosd(theta1+16)} {y30 = R_O*sind(theta1+16)}
point 32 {x32= R_I*cosd(theta1+16)} {y32 = R_I*sind(theta1+16)}

```

These points are best explained in the following figure:

Flooded-roll nip with film split



Point 22 is a point along the circular arc used to estimate the film split. Point 24 is the center of that circular arc as you can see it is used in a CIRC line type in **roll.fas**. Points 30 and 32 are a related position of this region of the mesh which describes the film split. To move the meniscus in or out of the nip, you change the phase angles on the points 22, 30 and 32 cards, keeping the phase angle of 30 and 32 greater than that of point 22, by about a couple of degrees. To position your meniscus further into the nip, increase all these phase angles by a few degrees, regenerate the mesh with **fastq**, and have a look at it. For splits that are WAY into the nip, you need to increase your curvature. You can accomplish this by moving point 24 to be weighted more heavily towards point 22 rather than 20, viz.

```
point 24 {x24 = (x22+x20)/2.} {y24 = (y23+y22)/2}
```

to

```
point 24 {x24 = (10.*x22+x20)/11.} {y24 = (y23+y22)/2}
```

First generate your mesh with the current conditions in the **roll.fas** file:

```
fastq -a -mesh roll.gen roll.fas (in batch mode, or you can do it interactively if you wish)
```

```
ex1ex2v2 roll.gen roll.exoII
```

Next get a fixed grid solution by making sure the “**Initial Guess**” card in the **roll_input** file is set to “**zero**”, and the KINEMATIC, CAPILLARY, and SURFTANG cards are commented out, e.g.,

```
$
$ Film Split Meniscus
$
BC = VELO_NORMAL SS 100 0.
$$BC = KINEMATIC SS 100 0.
$$BC = CAPILLARY SS 100 0.01 0 0
```

-9-February 29, 2000

Distribution

```
#BC = CAP_ENDFORCE NS 200 {sind(theta1)} {-cosd(theta1)} 0. 0.01
#BC = CAP_ENDFORCE NS 210 {sind(theta1)} {cosd(theta1)} 0. 0.01
#BC = CAP_ENDFORCE_SCALAR NS 200 -0.01
#BC = CAP_ENDFORCE_SCALAR NS 210 0.01
```

Also, make sure that the augmenting condition volume constraint is turned off, viz. the section

```
-----
Augmenting Conditions Specifications
-----
Number of augmenting conditions = -1
$$AC = VC {mat_id = 1} {volid = 1} {bcid = 11} {dfid=0} {compid = 0}
{const =2.2
03709e-03}
```

should not be active, as is accomplished here with the \$\$ comment on the AC line. Please see the advanced capabilities manual ([SAND2000-2465](#)) for details here.

Run GOMA: `goma -a -i roll_input`

It is really important that you look at the fixed grid solution and make sure that it is smooth. We have noticed that around the presumed film split region, solving for the fixed but slippery surface with the VELO_NORMAL boundary condition can lead to wiggly discontinuous velocity fields along the surface. You will NEVER converge to a free surface solution if this is the case. We find in roll coating that this initial fixed grid solution can be made to be smooth by just specifying an inlet flow rate, by assuming your initial film thickness guess and roll speed determines what the flowrate should be. You will notice the following BCs:

```
$$ Inflow boundary (specified pressure or velocity)
$$BC = FLOW_PRESSURE SS 5 0.
BC = U NS 5 {h_t*rollsp_b*2/(y15-y5)}
BC = V NS 5 0.
```

We find it necessary to start with a specified inflow velocity for the fixed grid solution step above. In fact, we could not converge by simply getting a solution for any arbitrary pressure with the fixed grid once the free surface was release. The inflow velocity on nset 5 here is adjusted so that the flow rate is matched with the outflowing films, assuming those films are in solid body rotation at the chosen film thickness and roll speed.

Now we are ready to go after a free surface solution. In the past we had the user go through a “pressure” inlet determination that went as follows:

```
$$ Inflow boundary (specified pressure or velocity)
BC = FLOW_PRESSURE SS 5 0.
$$BC = U NS 5 {h_t*rollsp_b*2/(y15-y5)}
$$BC = V NS 5 0.
```

Notice here we have turned off the specified velocity at the inflow, used to generate the fixed grid solution, and turned on a specified pressure condition. Here we arbitrarily set it to zero.

Now we have the convenience of solving for the inlet pressure with an augmenting volume constraint. To do this we first need to determine the volume. First prepare your input deck as follows:

Make sure BEFORE you do this you change the “**Initial Guess**” card in the `roll_input` file to “**read**” and change the free surface conditions back,

```
$ Film Split Meniscus
$
$$BC = VELO_NORMAL SS 100 0.
BC = KINEMATIC SS 100 0.
BC = CAPILLARY SS 100 0.01 0 0 0
BC = CAP_ENDFORCE NS 200 {sind(theta1)} {-cosd(theta1)} 0. 0.01
BC = CAP_ENDFORCE NS 210 {sind(theta1)} {cosd(theta1)} 0. 0.01
#BC = CAP_ENDFORCE_SCALAR NS 200 -0.01
#BC = CAP_ENDFORCE_SCALAR NS 210 0.01
```

Also, uncomment the AC condition, viz.

```
-----
Augmenting Conditions Specifications
-----
Number of augmenting conditions = -1
AC = VC {mat_id = 1} {valid = 1} {bcid = 11} {dfid=0} {compid = 0} {const
=2.203709e-03}
```

Run one iteration with zero relaxation to determine the current volume of the mesh. That is,

```
goma -a -i roll_input -n 1 -r 0.0
```

GOMA will give back the following

```
-----
Augmenting Conditions:      1
Number of extra unknowns:   1

MT[ 1] VC[ 1]=2.203709e-03 Param=3.264392e+03
```

So, you now want to make sure that the last floating point on the AC=VC record corresponds to the volume of the mesh, which here is 2.203709e-03. You can now run a full simulation with a little initial relaxation to get the steady state free surface solution.

```
goma -a -i roll_input -r 0.1 -n 10
cp soln.dat contin.dat
goma -a -i roll_input
```

You can definitely see the convenience of the volume constraint capability, particularly if you compare this approach with what used to be the case (cf. this extract from a previous version of this memo below):

-----OLD WAY-----

Note that on the fixed grid solution that if you plot the pressure and zoom in on the meniscus region that the range of the pressure goes from around 0 to 10 around the film split. Notice the pressure at the film split is about 7 Pa. For a surface tension of 60 dyn/cm, the capillary pressure at that point is ROUGHLY $(0.06 \text{ N/m}) / (0.195 - 0.189 \text{ m}) / 2 = 5 \text{ Pa}$ (N.B. the coordinates or length scale is picked off of the axis when zoomed in with **blot**). These are not quite matched (i.e., the pressure on the liquid side of the high curvature part of the meniscus should be 5 Pa due to the pressure jump), but probably close enough. Obviously the meniscus will seek a higher pressure region upstream to balance with and tighten its curvature so that it can balance the hydrodynamic pressure, in the low Ca number limit. The Capillary number here is roughly $(0.05 \text{ rad/s}) * (2 * R_O * \text{Pi m/rad}) * (0.1 \text{ Pa-s}) / (0.06 \text{ N/m}) = 0.1$. Actually this is quite high, but it still works. Also, the higher the Ca, the more the meniscus tends to get sucked into the gap, but viscous forces are more important, and potentially inertia too, so one must be careful in this regime in using this simple analysis.

-----END OF OLD WAY-----

Changing the roll separation:

For fun you can change the roll separation from 1 mm to 2 mm by changing **Gap_new** in **roll.fas** from 0.001 to 0.002 with the following sequence (or just type “**source run2**”):

```
cp soln.dat contin.dat
goma -a -i roll_input -r 0.1 -n 10
cp soln.dat contin.dat
goma -a -i roll_input -r 0.5 -n 10
cp soln.dat contin.dat
goma -a -i roll_input
```

Notice when looking at this solution that you have widened the gap with a specified pressure, so the pressure drop changes significantly and the meniscus nearly gets blown out of the domain. In this case, unlike the deformable roll case above, you will notice that changing the gap we are actually changing the equation of the circular line on side set 2 through the following BCs:

```
BC = GD_PARAB SS 2 R_MESH_NORMAL 0 MESH_POSITION2 0 {x1*x1 + y1*y1 - R_O_new*R_O_new} {-2.*y1} 1
BC = GD_PARAB SS 2 R_MESH_NORMAL 0 MESH_POSITION1 0 {0.} {-2.*x1} 1
```

Please look at the manual to figure out what these mean. In a “nut-shell”, you can see they specify an equation of a circle with radius **R_O_new** centered about the point [x1, y1]. The are accumulative and there are some nice examples in the manual of how to read them. The **R_MESH_NORMAL** entry means that the equation replaces the normal component of the mesh equations along that side set. This is very different from the deformable case in which the roll surface was “free”.

Changing the roll radius:

Changing the upper and lower roll radii. Change **R_I** from $7 * 0.0254$ to $6.9 * 0.0254$, i.e.,

```
Roll Radius          {R_I = 7.*0.0254} {R_I_new = 6.8*0.0254}
```

Run **goma** with 10 iterations at **-r 0.1** and then full Newton. Notice here with **blot** that the gap changes as well (zoom in and you can see). So if you are increasing the radius, you must also account for the gap change.

DEBUGGING NOTES:

- If you ever get termination of GOMA with the following message, that means the mesh “blew up”. You should relax more or take a smaller parameter step change.

```
Volume change -0.126022
Deformation Gradient 4.865954 3.642131
Deformation Gradient 2.177972 -0.000544
```

- When you are unsure of what is going wrong when you repeatedly get this message or when the flow is not converging, it helps to turn the “**Write Intermediate Solution**” card to “**yes**” and take some relaxed (i.e., **-r 0.1** or **-r 0.01**) steps and view the results with **blot**. This option causes GOMA to dump all Newton iterations to the **exoII** file.

Reverse Roll Coating with a Flooded Upstream

In a directory next to the **film_split_rigid** directory under **roll**, you’ll find a **film_split_reverse** directory. Enter that directory and you will find the same set of files as in the forward roll case. This exercise will simply show you how a reverse roll coater was accomplished using the same mesh and the same model. Use the Unix **diff** command to compare the **roll.fas** file and **roll.input** file from the **film_split_rigid** case:

```
diff roll.fas ../film_split_rigid/roll.fas
```

gives

```
11,12c11,12
< $ bottom roll speed      {rollsp_b = -0.1}
< $ top roll speed         {rollsp_t = 0.2}
---
> $ bottom roll speed      {rollsp_b = 0.05}
> $ top roll speed         {rollsp_t = 0.05}
18c18
< $ Outer Roll Radius + 2.5*film thickness {R_O_2h = R_O + 1.2*h_t}
---
> $ Outer Roll Radius + 2.5*film thickness {R_O_2h = R_O + 2.5*h_t}
20c20
< $ Roll extent {theta1 = 70} {theta2 = 120}
---
> $ Roll extent {theta1 = 60} {theta2 = 120}
38c38
< point 22 {x22= R_O_h*cosd(theta1+8)} {y22 = R_O_2h*sind(theta1+8)}
---
> point 22 {x22= R_O_h*cosd(theta1+14)} {y22 = R_O_2h*sind(theta1+14)}
40,41c40,41
< point 30 {x30= R_O*cosd(theta1+10)} {y30 = R_O*sind(theta1+10)}
< point 32 {x32= R_I*cosd(theta1+10)} {y32 = R_I*sind(theta1+10)}
---
> point 30 {x30= R_O*cosd(theta1+16)} {y30 = R_O*sind(theta1+16)}
> point 32 {x32= R_I*cosd(theta1+16)} {y32 = R_I*sind(theta1+16)}
45d44
< {x20= R_O_h_new*cosd(theta1)} {y20=R_O_h_new*sind(theta1)}
63c62
< point 24 {x24 = (20.*x22+x20)/21.} {y24 = (y23+y22)/2}
---
> point 24 {x24 = (10*x22+x20)/11.} {y24 = (y23+y22)/2}
```

-13-February 29, 2000

Distribution

Note here that besides some of the operating conditions being different (most notably the lower roll being turned in the opposite direction as can be seen by the negative sign on the `roll_sp` variable), we had to, by trial and error, adjust the roll extent and the film-split meniscus location (points 22, 30 32) and the film split curvature (point 24). These are more-or-less experience moves, but are motivated by the same sort of capillary pressure balance exercise we went through in the for the forward roll case.

```
diff roll_input ../film_split_rigid/roll_input
```

gives

```
57,61c57,67
< $$BC = FLOW_PRESSURE SS 5 2484
< BC = U NS 5 {-h_t*rollsp_b/(y15-y5)}
< BC = V NS 5 0.
< BC = U NS 66 {rollsp_b*sind(theta1)}
< BC = V NS 66 {-rollsp_b*cosd(theta1)}
---
> BC = FLOW_PRESSURE SS 5 984
> $$BC = U NS 5 {h_t*rollsp_b*2/(y15-y5)}
> $$BC = V NS 5 0.
> $
> $ Film Split Meniscus
> $
> BC = VELO_NORMAL SS 100 0.
> $$BC = KINEMATIC SS 100 0.
> $$BC = CAPILLARY SS 100 0.01 0 0 0
> $$BC = CAP_ENDFORCE_SCALAR NS 200 0.06
> $$BC = CAP_ENDFORCE_SCALAR NS 210 0.06
63,69d68
< $$BC = VELO_NORMAL SS 100 0.
< BC = KINEMATIC SS 100 0.
< BC = CAPILLARY SS 100 0.03 0 0 0
< BC = CAP_ENDFORCE_SCALAR NS 210 0.03
< BC = DX NS 200 0.
< BC = DY NS 200 0.
```

The first difference you will notice regards the inlet conditions. We use the **FLOW_PRESSURE** inlet condition of 2484 Pa in the forward roll case, and a specified inflow in the reverse roll case. This is on NS and/or SS 5. The specified inflow is set to be the make-up from the inlet velocity on the reverse turning lower roll, as you will notice the the forward roll is turning at twice the rate of the reverse roll. The film thicknesses are initial predicted to be the same, but that is only as an initial guess and actually they turn out to be a factor of two different. Also notice that on NS 66 we now specify an inlet plug flow velocity. This node set is across the inlet film on the bottom reverse roll. Before in the forward roll case we had a fully developed condition (i.e, we did nothing). Also notice that we pin node set 200 in the reverse case, and use **CAP_ENDFORCE_SCALAR** in the forward case. This is because in the reverse case we are setting the flow rate at the roll inlet by setting the roll speed and the film thickness. Node set 210 still has a **CAP_ENDFORCE** condition as it is still an outlet film.

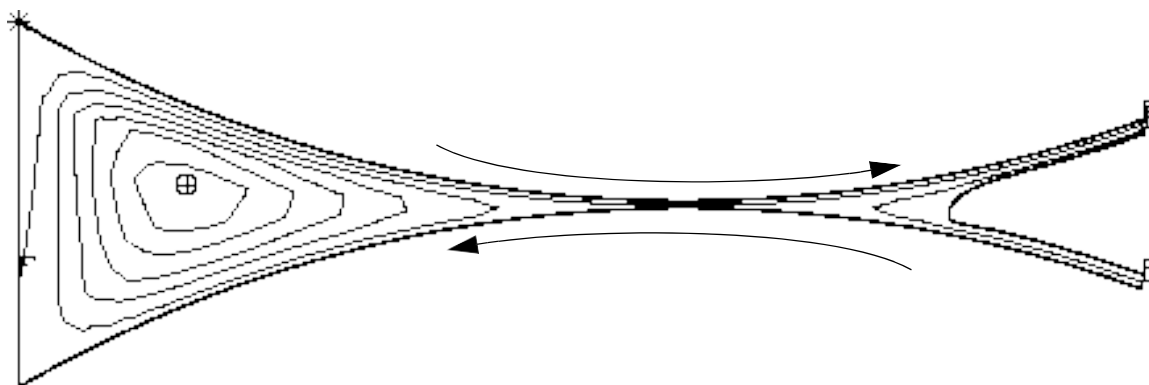
As in the forward case, obtain an initial solution by appropriately commenting out the **KINEMATIC**, **CAPILLARY**, and **CAP_ENDFORCE** cards on the free surface (SSET 100) and applying the **VELO_NORMAL** condition there. Set the initial guess to “zero”. Run **goma**:

```
goma -a -i roll_input
```

Now copy the solution file into the continuation file (`cp soln.dat contin.dat`), set the Initial Guess card to “`read`”, and change out the `VELO_NORMAL` for the `KINEMATIC`, `CAPILLARY`, and `CAP_ENDFORCE` cards, as before. But before we let this one go, please look at the pressure contours in with blot in the “`out.exoII`” file. Notice that the pressure at the assumed film split location is 38 Pa. Zooming in on this and estimating the curvature results in a capillary pressure of 30 Pa (i.e. $0.06/(\cdot 1925 \cdot 1900)$). So we may be in good shape. Then let rip the following sequence:

```
cp soln.dat contin.dat
goma -a -i roll_input -r 0.05 -n 20
cp soln.dat contin.dat
goma -a -i roll_input
```

You can look at the results and then try several other things. For instance you can change out the flowrate inlet condition for a reasonable pressure condition, or you can do the same gap and radius changes as in the forward case. The “take-home” lesson again here is the pressure balance approach to getting a good initial guess. In this case I changed the position of the split with the points 22, 30 and 32, and the curvature with point 24, as explained above. The solution looks like:



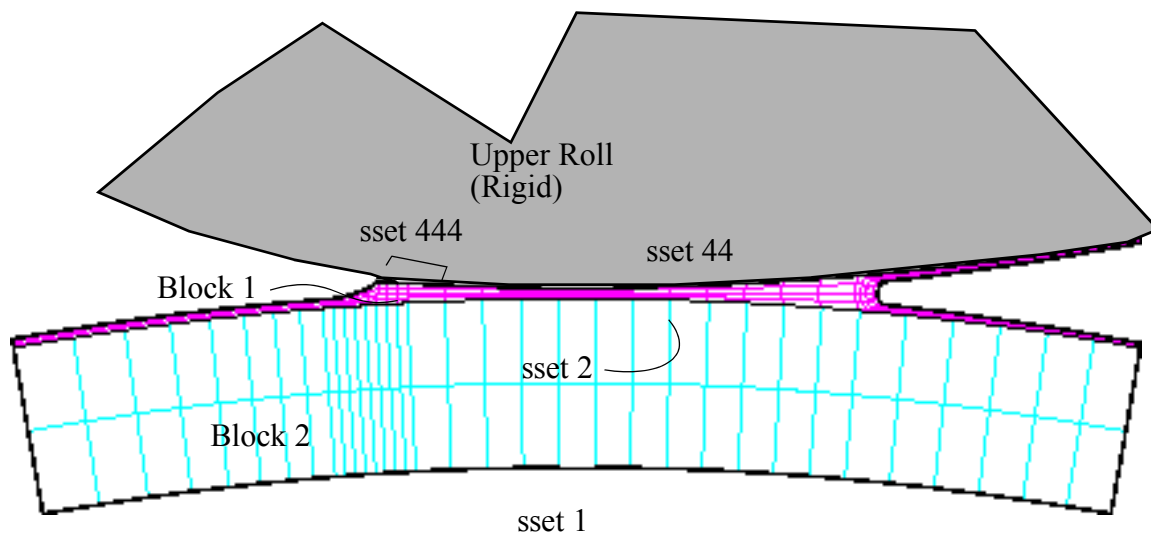
As in the previous example, it would be expedient here to deploy a volume constraint to help get the initial free surface solution. Just now on the left and outflow/inflow plane we actually specify the flow rate. This is not realistic and it would be better to use `FLOW_PRESSURE` here, with the AC volume constraint to determine the pressure. The user, if he/she wants to pursue this model in more detail, should pursue this improvement.

Full Forward Roll Coating

The geometry and the defining `fastq` entities for this problem are for the most part the same as those used in the CRMPC TALE algorithm tutorial memo (GT-005.3). In that memo, we carry out the analyses very similar to the one here, but with a deformable roll. It is highly advisable that the user

master that memo before attempting this example, if you desire to pursue a full roll coating model with a deformable roll. This example will consider the rigid roll case. What is added here are some of the developers' experiences in getting an initial guess for realistic coating conditions, a new continuation scheme that helps you get to high coating speeds and large compressions (together with a newly designed mesh), and the additional features needed for extension of this model from two rigid rolls to one deformable and one rigid roll. We have updated this from its predecessor GT-003.1 with additional features that help the user get the original solution. The files which pertain to this example can be found in **forward.9/rigid-roll**.

The geometry and conditions in this problem is illustrated here



Conditions: Lower Roll Speed 150 cm/s; Upper Roll Speed 150 cm/s; Gap = 0.02 cm

In this memo we will not consider the deformable roll, and so there is only one material block, 1. We will attempt to get a solution with the following conditions:

TITLE

Forward Roll coating with rolling back and film split

\$ GEOMETRY AND OPERATING CONDITIONS (CGS)

\$ gap (leading edge to substrate) {Gap = 0.02} {Gap_new = 0.020}

\$ roll cover thickness (1" = 2.54) {d = 1.25/4.}

\$ initial film thickness for $v_2/v_1=1$ {h_t = 0.012/2.}

\$ Inlet film thickness {h_I = 0.012} {h_I_new = 0.012}

\$ Roll Radius {R_I = 5.7} {R_I_new = 5.7}

\$ Outer Roll Radius {R_O = R_I + d} {R_O_new = R_I_new + d}

\$ Outer Roll Radius + film thickness {R_O_h = R_O + h_t} {R_O_h_new = R_O_new + h_t}

\$ Outer Roll Radius + 2.5*film thickness {R_O_2h = R_O + 1.9*h_t}

Distribution

```
$ bottom roll speed {rollsp_b = 150} {rollsp_b_new = 150}
$ bottom roll speed {rollsp_t = 150.} {rollsp_t_new = 150.}

$ Roll extent {theta1 = 82.5} {theta2 = 91}
$ Extent of inlet region film, in degrees: {theta_inlet = 4}
```

Notice here that the conditions are such that the line speed is 150 cm/s (*N.B. all previous roll coating templates have been in MKS, so be aware*). The way in which the lower roll speed is set is different than in the deformable case. Here we set the lower roll speed on the boundary and NOT through an advective Lagrangian velocity card in the material files. Also notice that the gap is 0.02 cm and the hard roll radius is 5.7 cm. In order to expedite the location of the film split, we will start with a zero density in the **liquid.mat** file. Actually it is set at 0.05 g/cm³. It should be of the order of 1 g/cm³. Just as well you could turn off the advective terms with the term multipliers for the liquid phase in **roll_input**. We will seek to turn the inertia terms back on once we have a solution. *It is noteworthy that the approach here is only the author's experience. Surely there are many other approaches to the ultimate state. The purpose here is to catalog an experience that will enable the user to learn and invent independent techniques.*

Several things to note about the mesh being used here. It has been made up of mainly structured regions for good reason. If one chooses to pave the liquid regions, this typically translates to a much higher density of mesh. That density has to be carried through the solid roll and becomes inefficient there. As in the rigid roll case, the key to this problem is locating the film split and rolling bank. The following sections of **roll.fas** help you do that:

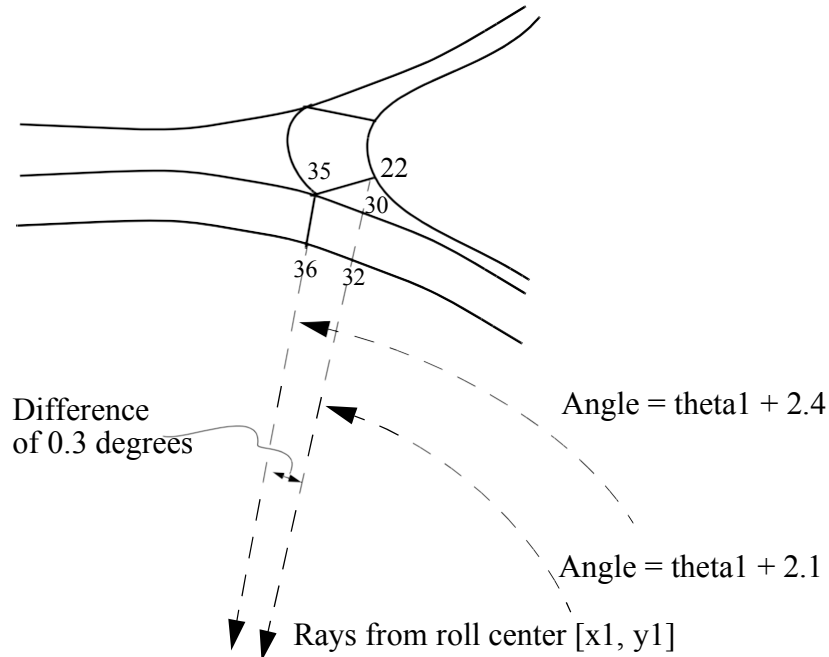
```
$$$$$$$The offset angles here determine the location of the film split$$$$$$
$$$$$$$Keep the phase angles the same for points 22, 30, 32. Same for$$$$$$
$$$$$$$Same for points 35 and 36. The difference is the length of the$$$$$$
$$$$$$$mesh region that resolves the film split.          $$$$$$
point 22 {x22=R_O_h*cosd(theta1+2.1)} {y22 = R_O_2h*sind(theta1+2.1)}

point 30 {x30=R_O*cosd(theta1+2.1)} {y30 = R_O*sind(theta1+2.1)}
point 32 {x32=R_I*cosd(theta1+2.1)} {y32 = R_I*sind(theta1+2.1)}

point 35 {x35=R_O*cosd(theta1+2.4)} {y35 = R_O*sind(theta1+2.4)}
point 36 {x36=R_I*cosd(theta1+2.4)} {y36 = R_I*sind(theta1+2.4)}
```

These points describe the film split region shown below. The key is that the points defining this region are on the lower roll and associated with the angles and phase offsets specified on the lower roll only. Since the original assumed film split is symmetric about the centerline, the upper points (mirrored about that center line down the middle of the rolls) are simply computed as dependents on these points. The angle **theta1** is the right most extent of the roll, as measured from the right pointing horizontal axis. You can see that the film split is assumed to be 2.1 degrees in from that outflow, and that the mesh around the film split is at 2.4 degrees offset. If you widen the gap between these angles, the four sided region centered on the film split will simply elongate. Experimenting with these angles and simply re-running **fastq** will show you how it works.

The rolling bank points are more complex.



The rolling bank (or upstream meniscus) is characterized by the following points:

\$\$ These points are to add the rolling bank and inlet region

```

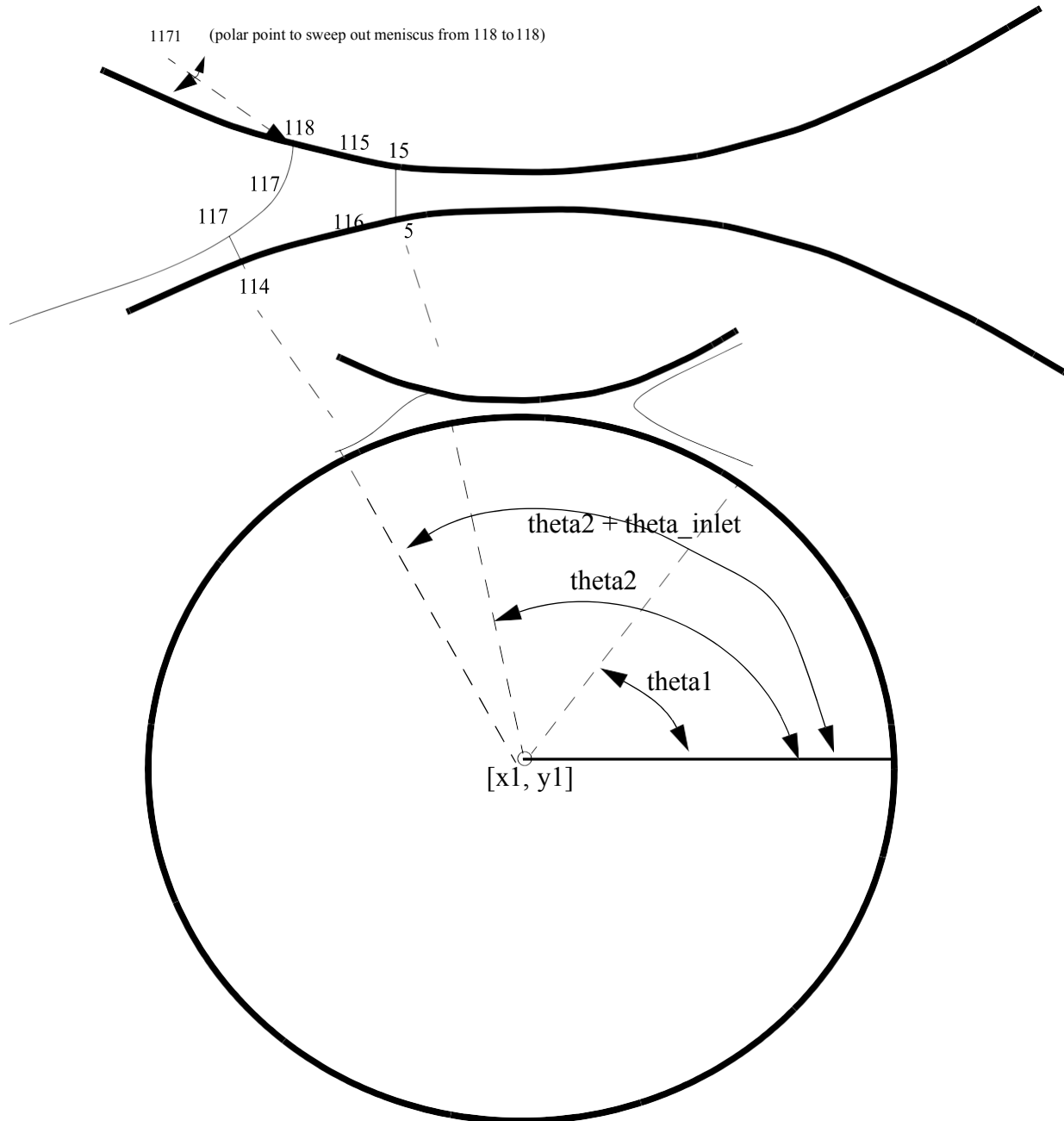
point 114 {x114 = (R_O)*cosd(theta2+1.0)} {y114=(R_O)*sind(theta2+1.0)}
point 1141 {x1141 = (R_I)*cosd(theta2+1.0)} {y1141=(R_I)*sind(theta2+1.0)}
point 115 {x115 =R_O*cosd(theta2 + theta_inlet)} {y115=R_O*sind(theta2+theta_inlet)}
point 1151 {x1151 =R_I*cosd(theta2 + theta_inlet)} {y1151=R_I*sind(theta2+theta_inlet)}
point 116 {x116 = (R_O+h_I)*cosd(theta2 + theta_inlet)} {y116=(R_O+h_I)*sind(theta2+theta_inlet)}
point 117 {x117 = (R_O+h_I)*cosd(theta2+1.0)} {y117=(R_O+h_I)*sind(theta2+1.0)}
point 1171 {x1171 = (R_I+4.*d/5)*cosd(theta2+1.0)} {y1171 = 2.*ym - (R_I+4.*d/5)*sind(theta2+1.0)}
point 118 {x118 = (R_O+h_I)*cosd(theta2+0.7)} {y118 = 2.*ym - (R_O)*sind(theta2+0.7)}

```

To help clear things up, consider the figure below. The points associated with the rolling bank are positioned relative to θ_2 and some offsets, just as those associated with the film split were placed around θ_1 . The figure helps clarify how all are related. In the case shown here, the offset angle for points 114, 1141 and 117 is 1.0 degrees, i.e., these points are an additional 1.0 degrees beyond θ_2 . Point 118 is 0.7 degrees beyond θ_2 . So if you want to locate the rolling bank further out of the nip to the left, but keep its rough size, simply increase both offset angles appropriately. Likewise if you want to place it further into the nip, you simply decrease them appropriately.

Our approach to the full solution can be summarized as follows:

1. Even those since the original release of this template we've developed many tools which aid in obtaining a base case solution, e.g. volume constraints, etc., you still need to do some preliminary work to have a chance at finding your way into a realistic operating space. Find a paper, thesis, or experimental data which reflects a roll coating situation under realistic speeds and conditions AND that you know had a stable, realizable solution. We used the Ph.D. thesis



by Dean Benjamin (“Roll Coating Flows and Multiple Roll Systems”, 1994, U. Minnesota, available on University Microfilms, Ann Arbor, MI). You have got to get into the “ballpark” on this problem or you will never make it in a reasonable time. Solutions for this flow don’t exist at a widespread set of parameters. From this thesis I gleaned some the base conditions. Unfortunately in that thesis as in some others, all results were for zero Reynolds number, and it is difficult to figure out the “extent” of the bead because they magnify the y-axis and don’t tell the reader what the actual coordinates are; however, in some plots you can get an idea. Moreover, they work in dimensionless parameters, so you need to analyze everything carefully.

2. Do a simple pressure profile analysis to help locate the film split. This is the most critical part of the process. Although in this tutorial we will use a volume constraint to help us obtain the solution, this analysis is helpful for a general understanding. If you get the film split converged, you are more-or-less “home free”. A good example of the meaning of the pressure profile is to look at the viscocapillary analyses that abound in works coming out of UMN. Perhaps you have your own mini-codes that evaluate the same things. Now several papers/thesis discuss where film splits locate, in relation to the pressure profile along the rolls, in various parameter ranges. The upstream pressure/flow rate condition and many other factors influence this position. Benjamin, Coyle, and others suggest that at low capillary numbers the best approach is to use the simple dip coating flow analysis by Landau and Levich (1942, “Dragging of a liquid by a moving plate. Acta Physicochim. USSR 17:42).
3. Plot the pressure profiles using the new data printing option (illustrated below). You can also plot this using the `user_print.c` option. GOMA’s distribution has a commented out example of how to do this in `user_print.c` (see Appendix on how to activate this). You will have to rebuild GOMA to use this. You can also just pick off the node numbers along the lower roll in blot, or a sample, and use the `splot` capability in blot to get the profile. I find it help to get this profile. These plots (see below) allow you to get an educated guess at the position.
4. Adjust the `theta1` and `theta2` parameters in `roll.fas`, together with the offsets described above in the same file, to get the presumed location in the right area. Run a “fixed” grid solution and look at the pressure profile. What I find useful here is described below.
5. Experiment with initial guess locations by adjusting the angle offsets above around the film split, the presumed curvature (by adjusting the weightings on point 24), viz.

```

$$ Define center for arc of circle on film split
point 24 {x24 = (16*x22+x20)/17; {y24 = (y23+y22)/2}

```

\$ here is you raise the 16 to 26 and the 17 to 27, you get a much higher curvature. Also, you can adjust the slopes down to the final film thickness by adjusting the following card in `roll.fas`:

```

$ Outer Roll Radius + 2.5*film thickness {R_O_2h = R_O + 1.9*h_t}

```

\$ so that the 1.9 is higher or smaller.

6. Run the “`run1`” script or use conservative relaxation, until you converge. This part takes some patience. I am lucky sometimes and unlucky others. Don’t get locked into a “mind set” that the solution HAS to be here. On the other hand, be patient on the small relaxation factors. Every case is different. I have been able to converge with 10 iterations of 0.1 followed by full Newton in some cases, and in others it takes 50 or more iterations at 0.005 before you can start increasing the relaxation parameter towards 1. Typically you can observe where the split is “trying” to go, and adjust your geometrical guess appropriately. N.B. EVERY TIME YOU CHANGE A GEOMETRICAL PARAMETER, MAKE SURE YOU REGENERATE YOUR MESH WITH FASTQ AND EX1EX2V2 BEFORE TRYING AGAIN. THIS MEANS YOU OF COURSE HAVE TO GET A FIXED GRID SOLUTION AGAIN. I am continually amazed how touchy this step is. You can play games with the backpressures or volume constraints, but then you are stuck with continuation in a parameter back to its normal state. We will show some examples of this below.
7. The upstream meniscus is relatively easy once you have converged on the downstream meniscus. You now must fix the flow rate, as shown below, and sometimes add a little back

pressure on the capillary card to get it to converge. Once both surfaces are released you can add inertia slowly, compress the rolls, lower the flow rate, etc. You can also reduce your artificial backpressure to the real solution. So now, read the procedure below for this one case, and learn how to get your pressure profiles along the nip out. Some other noteworthy cause-and-effects that can be gleaned from papers, common sense, or by trial-and-error:

- Increasing inertia (i.e., by raising liquid density, for instance) pushes film split out and rolling bank out
- Decreasing gap sucks in split but pushes out rolling bank.
- Decreasing inlet film thickness sucks in split and pulls rolling bank into nip.

So here is the experience I had in getting to the following conditions: Roll Speeds (both): 150 cm/s; Liquid density $1 \text{ cm}^3/\text{s}$; roll separation 0.01 cm; roll hardness ($1.e9 \text{ dyn}/\text{cm}^2$ Lamé and $1.e9 \text{ dyn}/\text{cm}^2$ Mu, slightly compressible); Liquid viscosity 0.1 poise; Roll radii 5.7 cm, lower one with a 1.25/4 cm cover; and incoming film thickness 0.009 cm. These for the most part are the main parameters. I ran out of time but was one the way to a negative compression, but clearly I don't know what the operation conditions are. What you will see here is that I was on the way through parameter space in the narrow coating window. I am confident you can get to most stable conditions with this approach. Here we go.

Now, with all the mesh details and the general approach out of the way, lets go after a solution to the fixed grid case and the film split. Queued up are the files **roll.fas.fixed**, **roll_input.fixed**, and **liquid.mat.fixed**. Copy these files appropriately into the usual

```
roll.fas
roll_input
liquid.mat
lower_roll_cgs.mat
```

This is already set for the fixed grid solution. The chosen film split location was based on the force balance analysis below. Before we tackle that analysis, get a fixed grid solution, viz.

```
goma -a -i roll_input -r 1.0
```

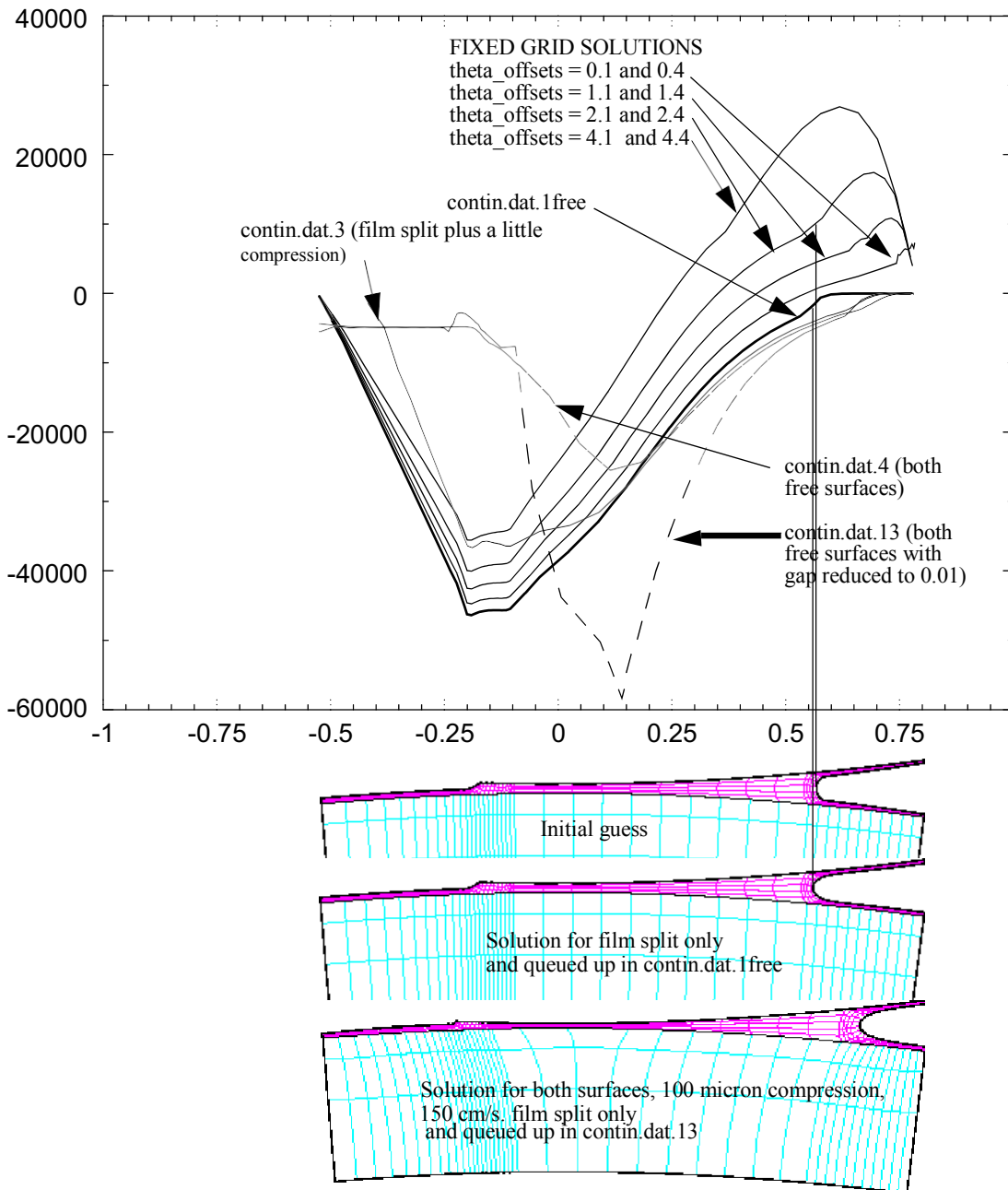
Notice here in the **roll_input** file that the inflow velocity at side set 111 is not fixed:

```
$inflow planes
$Note the adjustment on the inflow speed level for the fact that
$rollsp_b is in rad/s and we need cm/s.
BC = PLANE SS 111 {-tand(theta2+theta_inlet)} 1. 0. {-y1}
$$BC = U NS 111 {rollsp_b*R_O*sind(theta2+theta_inlet)}
$$BC = V NS 111 {-rollsp_b*R_O*cosd(theta2+theta_inlet)}
$$BC = FIX NS 111 VELOCITY1 0
$$BC = FIX NS 111 VELOCITY2 0
```

No velocity components are set there. This must be done in order to achieve the appropriate pressure profile on which to base you initial guess for the position of the film split. Now go ahead and “**plot out.exoII**” and contour first the variable “**pressure**”.

Distribution

Let us look at the pressure profiles for fixed grid case, at several different presumed film split locations. These are shown in the figure below. We actually show them for the deformable roll case,



but the same concepts apply here for the rigid roll. To get these profiles we turned on the post-processing data print option, viz.

```

Post Processing Data = -1
DATA = PRESSURE 2 1 0 pressure.out
END OF DATA
    
```

This card says that along node set 2, from material block ID 1 (the only one in this model) we will write out the primitive pressure degrees of freedom into the file **pressure.out**. In this file you will find the 3 coordinates of each node on the side set, and the corresponding pressure. You can plot those with gnuplot or your favorite spreadsheet program.

The previous edition of this memo, viz. GT-003.0, went through extensive balance-of-force arguments as to how you should choose your initial film-split location. Below we leave those arguments for you to read and learn with, but with the new Volume constraint capability discussed in [SAND2000-2465](#), there is no need to go through that complexity.

-----Aside: old way of determining film split location for the initial guess-----

You can see that we varied the film split location and computed solutions for 4 different locations of the film split, and plotted the pressure profile. In the fixed grid case you can immediately see why the force balance approach will not work, as the pressure profile along the nip is influenced by the assumed film split region, as the pressure rises due to the divergence of the nip and the impact of the fluid with the fixed film split, and then falls with the viscous pressure drop down the individual films. There is no surface tension force active here. So balancing these forces in a fixed grid case to generate a “good enough” guess for the free case will not work at high speeds. At low capillary number they do work because surface tension dominates viscous forces and the pressures will balance due to the relatively insignificant pressure drops (cf. previous memo on forward roll coating which was for a very low Capillary number case). The information that this little exercise provides is that as you vary the film split location in your fixed grid analysis, and look at the pressure profiles, the split will locate somewhere on the final up-slope of the pressure profile near the end. You will notice that with all of the offset angle cases, you can pin the location to within the range 0.5 cm and 0.6 cm. The best thing to do then is just start trying different locations.

After trying to interpret these profiles, we can tell you that the assumed film split offset angles of 2.1 and 2.4 (see geometry description above) will converge to a solution, and the rest will not. The fixed grid “guess” and the resulting film split solution can be compared here. Noticed that after the film split is released, the pressure downstream of the nip asymptotes back to zero, of course, rather than going through a peak. The guessed location is about 0.55 cm and the actual location is at about 0.548 cm. You can see that this is a pretty good initial guess. Unfortunately, that is what this very nonlinear problem requires.

-----End aside-----

Go ahead and obtain the film split solution with the first presumed film split location above. To do this you must uncomment the **KINEMATIC**, **CAPILLARY**, and **CAP_ENDFORCE** cards on and associated with side set 100, set the initial guess to read, make sure you copy your fixed grid restart solution **soln.dat** into **contin.dat**, AND, make sure that no flow rate is specified on side set 111. Moreover, go ahead and uncomment the **FLOW_PRESSURE** boundary condition on side set 111. We are going to use that pressure as an unknown for the volume constraint. Also comment out the **VELO_NORMAL** card on SS 100. Finally, uncomment the **AC = VC** card in the augmenting condition section, viz.

Distribution

Augmenting Conditions Specifications

```

-----
Number of augmenting conditions = -1
AC = VC {mat_id = 1} {valid = 1} {bcid = 11} {dfid=0} {compid = 0} {const
= 3.288167e-02 }
END OF AC

```

If you use the Unix “diff” command on `roll_input.fixed` with `roll_input.1free`, you will see the changes required to go from the fixed to the single free surface case.. In fact, you can just copy `roll_input.1free` into `roll_input` and `roll.fas.1free` into `roll.fas` and `contin.dat.1free` into `contin.dat`. To run this solution, just follow the following procedure:

```

goma -a -i roll_input.1free -r 0.0 -n 1
cp soln.dat contin.dat

```

The output from this run will look like

```

-----
Augmenting Conditions:      1
Number of extra unknowns:  1

      MT[  1] VC[  1]=3.288179e-02 Param=1.123583e-01

```

So go ahead and set the volume on the AC record to this value, as is shown above. Now go ahead and run with full Newton iteration,

```

goma -a -i roll_input.1free -r 1.0

cp soln.dat contin.dat

```

You can see that with the volume constraint we don’t even need to relax. Once you have the film split solution, the rest is much easier, as long as you are patient.

At this point we have not fixed the flow rate. Our upstream condition on the film feed is a constant, zero pressure condition. In a finite element formulation that is equivalent to the “no boundary condition” case, which really means you are applying a zero total stress condition which for the fully developed case implies zero pressure. At this point, to go onto to release the upstream meniscus I find it better to switch over to the fixed inflow rate.

Now we will release the upstream meniscus. Several things need to be done here. First, fix the flowrate at its current state by uncommenting the “FIX” BC cards:

```

$inflow planes
$Note the adjustment on the inflow speed level for the fact that
$rollsp_b is in rad/s and we need cm/s.
BC = PLANE SS 111 {-tand(theta2+theta_inlet)} 1. 0. {-y1_new}
$$BC = U NS 111 {rollsp_b*R_O*sind(theta2+theta_inlet)}
$$BC = V NS 111 {-rollsp_b*R_O*cosd(theta2+theta_inlet)}
BC = FIX NS 111 VELOCITY1 0
BC = FIX NS 111 VELOCITY2 0

```

Note here that on node set 111 we can either fix the velocity to be the speed and direction of the lower roll at that point, or we can “FIX” the components at the previously computed values. To keep the flowrate exactly the same, do the latter. Uncomment the BC=FIX cards as shown.

Now we are ready to release the upstream meniscus. The appropriate changes to the input file are already queued up in `roll_input.2free`. A Unix “diff” with `roll_input.1free` will show that the FIX commands above and the cards described below are the only changes. That is, to release the rolling bank, in addition to the FIX commands above, uncomment the CAPILLARY, KINEMATIC and CA cards associated with the upstream meniscus, side set 112, and comment out the VELO_NORMAL card, as usual:

```

.$Upstream Meniscus
$$BC = VELO_NORMAL SS 112 0.
BC = DX NS 220 {x116_new - x116} 1.0
BC = DY NS 220 {(y116_new - y116) + (Gap - Gap_new)} 1.0
BC = U NS 230 0.
BC = V NS 230 0.
BC = KINEMATIC SS 112 0.
BC = CAPILLARY SS 112 60. -23990. 0 0
BC = CA    NS 230 .855 0 1 0

```

Notice that the surface tension is 60 dyn/cm and the contact angle is 0.855 radians. More importantly the backpressure on the CAPILLARY card is set to -23990 dyn/cm² which is needed to keep everything in the nip, otherwise you notice it wants to blow out to the right. In previous edition of this tutorial this value was found by some trial and error. We still recommend you do the same. With these settings you can get a full free surface solution with the “run2” script, which contains the necessary relaxation strategy to get a solution, viz.

```

cp soln.dat contin.dat
source run2

```

-----Aside-----

You will notice the “commented out” volume constraint section in `rol_input.2free`:

```

-----
Augmenting Conditions Specifications
-----
Number of augmenting conditions = -1
$$AC = VC    {mat_id = 1} {volid = 1} {bcid = 26} {dfid=1} {compid = 0}
{const = 3.288167e-02}
END OF AC

```

Notice here that the bcid integer is set to boundary condition ID 26, because the capillary card applied to side set 112 is the 27th boundary condition in the input file (the first boundary condition card corresponding to zero). You can determine this number by changing SS to SC on this card and running GOMA with the `-bc_list` option (see Advanced Capabilities Manual for a complete description, SAND2000-2465). Also note that the float id (`dfid`) is set to one, as the backpressure on the upstream meniscus is set here as the second float of the CAPILLARY card (See GOMA User’s manual). We know the volume should be the same as the previous run, so we don’t change this. You

Distribution

-25-February 29, 2000

can try to get a solution with this constraint, and that may help you with subsequent continuation. We were not able to get this to work for the upstream meniscus with our brief testing.

-----End Aside-----

The final step to get to a realistic solution is to add inertia to the system. Heretofore we have had the liquid density set to zero in “**liquid.mat**”. Increasing the density to the desired value, about 1 g/cm³, will tend to push both menisci out. In anticipation of this, it is helpful to pull the film split in a little by squeezing the rolls more.

Change **Gap_new** to 0.02 from 0.0197 and run 10 iterations at relaxation automated relaxation scheme set up in the input deck, followed by a forced full Newton iteration, i.e.,

```
goma -a -i roll_input -n 10
cp soln.dat contin.dat
```

Now change **Gap_new** to 0.0193 and follow the procedure:

```
goma -a -i roll_input
cp soln.dat contin.dat
```

Now change **Gap_new** to 0.0187 and follow the procedure:

```
goma -a -i roll_input
cp soln.dat contin.dat
goma -a -i roll_input -r 1.0
cp soln.dat contin.dat
```

Now change **Gap_new** to 0.0185 and follow the procedure:

```
goma -a -i roll_input
cp soln.dat contin.dat
goma -a -i roll_input -r 1.0
cp soln.dat contin.dat
```

Notice here that the upstream meniscus is growing out of the nip.

No start increasing the density. I went from 0.05 to 0.15 in **liquid.mat** with full Newton iteration:

In liquid.mat:

```
---Physical Properties
Density          = CONSTANT    0.15
```

Take a Newton step:

```
goma -a -i roll_input -r 1.0
cp soln.dat contin.dat
```

At this point I continued to add density, which tends to push the menisci out, and squeeze the rolls together, which tends to pull the film split in and the rolling bank out. Another “knob” I used is to reduce the inflow flow rate, by reducing the inflow film thickness parameter **h_I_new**.

-26-February 29, 2000

Distribution

I found that alternating these steps worked fine, as long as I was aware where every meniscus was. As you go through all of this, save solution files off to the side, so if your continuation fails, you can go back to a nearby point of departure. I was able to wiggle my way to less than 100 micron gap and full inertia, with no backpressure. What I think we are actually doing is wiggling through the coating window. Now all this should become easier with automated continuation and stability checks, which is coming. Even better when the volume constraint is available, in the next major release, as you can float the backpressure to maintain the volume, which should help achieve real conditions much faster. But I think before all of those fancy tools are invoked, when available, it is helpful to go through this experience to get a feel for the power and limitations of the model.

IMPORTANT WARNING: Using the volume constraint with back pressure on the upstream meniscus as the unknown does not seem to work very well. Additional work must be done with the full forward roll model to find out what parameter can be release to accommodate the constant volume constraint. Roll separation? Contact angle?

Also, you will notice in `roll_input.2free` there are some hunting conditions for multiparameter continuation in Gap. These cards exemplify how you can invoke automatic continuation in Gap, even though Gap affects at least 5 boundary conditions in the problem.