SLIDES INTRODUZIONE:

* Seguire il capitolo 1: ci sono molte applicazioni di modelli accoppiati free-flow con mezzo poroso, per esempio questo e questo. (5 righe fatte bene sulla soil salinization mi farebbero comodo).
* Ci concentriamo per esempio sull’evaporazione dal suolo in cui abbiamo due sottodomini e un modello accoppiato blabla, ci sono una moltitudine di fattori che la influenzano (aggiungere la radiazione) noi prendiamo la turbolenza e gli ostacoli.
* La turbolenza e le eddies vicino all’interfaccia sono importanti per i processi di scambio, lo scopo di questa tesi è migliorare il modello del free-flow. Nell’ambito di una discretizzazione coi volumi finiti è importante l’approssimazione utilizzata per la convezione, si sono implementati in dumux metodi TVD che danno il secondo ordine ed garantiscono la stabilità.

FREEFLOW

* Si parte dalle equazioni di Navier-Stokes per un fluido newtoniano incomprimibile, quindi in seguito quando si considera la turbolenza si utilizza un approccio di tipo RANS con equazioni mediate ed entra in gioco la visosità turbolenza tramite l’approssimazione di boussinesq per gli sforzi di reynolds. Ci accontentiamo del valore medio del campo di velocità, teniamo a mente che i risultati non sono il fluido come appare nella realtà
* Come modello di turbolenza utilizziamo k-omega, (Kolmogorov, Wilcox). Utilizza k, l’energia cinetica turbolenta ed omega, dissipazione specifica. Ci sono varie versioni del modello, noi in particolare utilizziamo queste correzioni con il limiter per omega e per la production. Non ci sono motivi precisi, è un modello di turbolenza derivato in maniera empirica. Nel nostro caso è meglio del k-epsilon perché abbiamo gli ostacoli al bordo e non vogliamo quindi usare una wall-law, ma integriamo le equazioni fino al viscous sublayer.

POROUS-MEDIUM FLOW

* To model the flow in the porous medium we use a REV scale approach, averaging quatities over Reference volumes, in order to filter out irregularieties of the pore scale but keeping heterogeneities of the large scales. Averaging NS we obtain incompressibility of the porous medium and the Darcy’s law (obtained experimentally blabla), but we prefer the Forchheimer’s law (higher Reynolds numbers, smooth transition to a quadratic drag. Spiegare un attimo le equazioni: K è il tensore di permeabilità e Cf è un numero adimensionale, preso nel nostro caso uguale a 0.55 ma ci sono studi a riguardo.

COUPLING CONDITIONS

* At the interface Gamma\_int we apply conditions that should approximate the local thermodynamic equilibrium, even if, since we are coupling two different model concepts, we won’t be able to achieve it rigorously. We need only the mechanical equilibrium because we are isothermal and single component, we impose the continuity of mass fluxes, and of normal momentum for which we see that we could obtain a jump in pressure. We impose the BJS condition for the tangential momentum as a slip boundary condition in the free-flow, the more the porous-medium is permeable , the more slip is allowed.

FINITE VOLUME DISCRETIZATION

* In order to discretize the equations we employ finite volumes (instead of finite elements because…) we use the staggered grid so that we do not have oscillations (?) and we have the velocity variables at the interface, it will be easier to perform the coupling with the porous medium. Funziona quindi così e così, ogni variabile ha il suo control volume.
* Discretizziamo le equazioni, tutti i termini si discretizzano in maniera naturale, abbiamo solo dei problemi per il termine convettivo. Prendiamo per esempio tale termine nell’equazione del momento, abbiamo una transporting velocity e una quantità trasportata. Per quella transporting effettuiamo una media, mentre per quella trasportata dobbiamo tenere conto della direzione del flusso. Nell’equazione di continuità abbiamo la densità costante quindi no problem, nell’equazione del momento la densità sempre può uscire dall’integrale, abbiamo però la velocità trasportata per la quale dobbiamo effetturare un’approssimazione
* Ci sono varie possibilità, quelle più standard sono per esempio un’upwind del prim’ordine, è una scelta facile e comoda, stabile, ma introduce molta viscosità numerica, alterando la soluzione smussando i gradienti. E’ comunque una buona scelta per avere un metodo robusto. Se si vuole più accuratezza bisogna usare metodi di ordine superiore: c’è il central differencing, che è del second’ordine ma non è stabile per numeri di peclet alti, e c’è il QUICK che è del terzo ordine, considera la direzione del flusso, però può generare overshoots o undershoots che possono portare a valori non fisici.
* Nell’immagine possiamo vedere il confronto tra questi metodi applicati ad un’equazione conservazione con solo trasporto in 1D.
* Vorremmo quindi mantenere la stabilità di un metodo come upwind ma aumentare l’accuratezza aumentando l’ordine di convergenza. Utilizziamo quindi metodi TVD: essi sono stati sviluppati inizialmente per equazioni iperboliche, consentono di avere un’accuratezza del secondo ordine e di non presentare alcuna oscillazione, essi hanno la proprietà di essere monotonicity preserving, cioè questo e questo. Essa è infatti ottenuta imponendo la condizione TVD, dove TV è definita così.
* Si procede aggiungendo al first order upwind un flusso antidiffusivo non lineare, il quale dovrebbe eliminare le oscillazioni quando presenti. Non lineare infatti era stato dimostrato che non cipuò essere uno schema lineare tvd di ordine superiore a 1. C’è psi che dipende da r, si impone la TVD condition e la second order, e la symmetry e si ottiene lo schema di sweeby.
* Rimandendo entro questi limiti si possono trovare molte funzioni flux limiters, come quelle riportate qua. In letteratura se ne possono trovare di continue, come Van Leer e Van Alabdada, e di lineari a tratti. Il minmod sta sul limite inferiore della regione mentre il superbee sta su quella inferiore. Tra queste non si riesce a dire a priori quale sia meglio delle altre, dipende da caso a caso. In generale però quando stanno vicino al limite superiore sono compressive, tendono quindi a creare gradienti più forti ma possono essere più problematiche nella convergenza, viceversa quando sono vicine al limite inferiore tendono a essere più smooth e a convergere meglio.
* Qui le vediamo rappresentate. Da sottolineare che sono definite uguali a zero per r < 0, quindi in presenza di estremi locali si utilizza solo il contributo di prim’ordine e si riduce l’accuratezza ad 1. Una proprietà che è richiesta ma non appare nello sweeby’s diagram is this symmetry property.
* We can see in the example of the 1D conservation law that with the TVD method the obtained result does not present any oscillation and it is as accurate as the QUICK.

NON-UNIFORM GRIDS

* What we have seen so far holds for uniform grids, but with non-uniform ones we must generalize the TVD approach to keep its desired properties. We exploit this also when, near a boundary with a Dirichlet condition for the velocity, we have three degrees of freedom that are not equidistant. In literature there are several different approaches.
* A first simple approach [Li, 2008] starts from the consideration that the TVD formula holds if the nodes D, U, FU are equidistant. If we are dealing with a grid in which they aren’t, we try to reconstruct a value for the velocity at the node FU\*, in such a way that FU\*, U, D are equidistant. This new value is constructed starting from the value FU and exploiting a linear approximation. A criticised aspect of this approach is that it does not consider the cell sizes.
* To improve that, another approach, more rigorous, was developed in [Hou, 2012]. With the aim of consider also the cell sizes, we start from a generalized TVD formula in with the factor small r is generalized considering not only the difference between the velocity but also the distances, so to have an approximation of the gradients. Then it is introduced a factor big R instead of the 2, which is defined as the ration between the sum of sizes of the upstream and downstream staggered cells, and the size of the upstream staggered cell. So, starting from this formula, we impose the TVD condition and the second order accuracy, obtaining a generalized TVD region. At last the flux limiter functions have to be generalized too in order to fit into this new diagram, but this operation can be done quite easily.

CELL-CENTRED DISCRETIZATION

* For what concerns the model in the porous medium we have only one scalar equation and we use the pressure as primary variable. We discretize it with a cell-centred approach, thus storing the degrees of freedom of the pressure at the centre of each cells and the cell itself correspond to the control volume. To compute the velocity we use the Forchheimer’s law, that is non-linear, so we need the Newton’s method to find the solution. For the viscosity and eventually the density we use and upwind approximation, while for the permeability we evaluate it at the surface or compute a harmonic average in case of discontinuous values, using thus the tpfa method.

TEMPORAL DISCRETIZATION

* For the temporal discretization want to use fully implicit method that guarantee the unconditional stability. For example, the backward Euler, that is of first order, or the BDF2, that is a two-step method of second order. In dumux the size of the time-step is usually chosen depending of the convergence of the Newton’s method at the previous time iteration, thus we have also generalized the BDF2 formula for a non-constant dt.
* The problem is finally solved at every time-step using the newton’s method with a numerically computed jacobian and with a monolithic approach using a direct solver (UMFPACK).

CONVERGENCE

* We have performed many convergence tests with respect to the size of the cells to see the behaviour of the error using a TVD method and using the first order upwind. For example, these are the results we have obtained with a steady problem for the NavierStokes equations over the unit cube, with Dirichlet boundary conditions. We have computed the L2 norm of the error against the analytical solution with Re=1000 and we have seen an increasement in the convergence order and a reduction in the absolute value of the error. With other problems we have observed non-optimal increasings, but however the results are always better than the upwind in particular at Re>>1.

ROUGH CHANNEL

* After that we have continued our comparison with a configuration nearer to our final goal, i.e. a flow in a channel with a flat wall on the upper boundary and a rough wall made of many small cavities in the lower boundary. We have computed a solution using the first order upwind and many TVD schemes and we have compared them against a solution computed using the upwind method but a very fine mesh (6 times finer), computing the L2-norm of the error along a section near the end of the channel, as we can see in the picture. We have obtained the all the TVD methods make better then the upwind with an error smaller of more or less one order.
* Then we tried to use deeper cavities, and to repeat the same comparison. This time difference is higher in the region of the cavity, as we can see from the plot. The solution of the upwind is smoother, moreover in the upper part of the cavity also the convexity of the plot is different.
* At last I solved the same problem of the first case but with Re=1 and we observed no difference. At this regime the convective term is not dominating.

BACKWARD FACING STEP

* Moving to the RANS equations we started with a very common test, the backward facing step, in order to compare our results with those available from a NASA report. In this configuration there is a channel flow with at a certain point a sudden enlargement. In the picture we can see a zoom over the area right after the enlargement and we can see the behaviour of the velocity field. The flow occurs into separation, there is a big recirculation, a corner one and a very small one at the corner. In this test an important indicator is the reattaching distance, i.e. the distance after the step at which the flow at the wall has a positive velocity. To measure this we have computed the friction coefficient C\_f and we have obtained a better approximation with the Van Leer, very near to the reference results, the upwind instead predicts too early the reattachment. There are also some experimental results from the 80’s, comparing the velocity profile the upwind seems in some situations to approximate better them, but here our results was to validate our model comparing it with known numerical results, so we are quite satisfied.

CAVITIES

* We want to study the coupling between a free-flow and a porous-medium with a shaped interface, that presents obstacles. As first approximation we use two little cavities on the lower boundary and we want to study of they influence each other. The flow field in the second cavity will probably change depending on the distance d between them. The flow field seems in the two cavities, we study the velocity profile of u along the section y = h. We see that the shape is the same but the values are a bit lower. The peaks are due to the fact that there is not the wall friction, then the sudden decrease due to the fact that there is a splitting in the velocity, so the u component decreases, then after a distance d\_rec it recovers its value. We report the maximum value of the velocity in the second peak and we see that below 0.5m the velocity is affected by the first one, while after 0.5 there is a slight decrease due to the development of the boundary layer and the increased turbulence created by the first cavity.

MULTIDOMAIN CAVITIES

* Finally, we switch to a multidomain simulation, with the space between the two cavities of the previous test filled with a porous medium, d=1. Coupling conditions on Gamma\_int and no flow conditions on Gamma\_base. We study this time how the two cavities affect each other depending on the permeability of the porous-medium. As before we compare the u velocity along the section y=h. We observe that for low values of permeability (<10^-8) the porous medium does not affect the flow field in the free-flow. Then increasing the permeability, we observe that in correspondence of the first cavity the maximum velocity increases, probably because the porous-medium reduces its wall effect since a more relevant amount of fluid enters in it. Then this fluid exits from the top of the porous-medium within the first 20cm, this fact spreads and moves ahead this zone with a peck in the y component of the velocity thus the more important decrease at the begin of the porous medium. With high permeabilities the velocity is restored to a higher value because of the BJS condition that allows the slip thus modifying the boundary layer profile. Eventually the behaviour around the second cavity is the same for all the permeabilities, because the corner eddy has low velocity its interaction with the porous-medium does not affect the rest of the system.

Good morning to everybody, today I will talk about the work of my thesis entitled a fully implicit higher order staggered grid formulation of the navierstokes equations for coupled flow systems. Coupled free-flow and porous media systems are common in many different applications, for example we can consider the case of an evaporation process from soil. As we can see in the picture we have the soil, that is a porous medium, and the air, a free-flow and they are separated by an interface, that is considered to be sharp. There can be many physical phenomena acting in a system like this, in this thesis I considered the effect of turbulence in the free-flow, also in presence of heterogeneities at the interfaces, such as roughness or obstacles. In such a situation turbulent eddies develop near the boundary