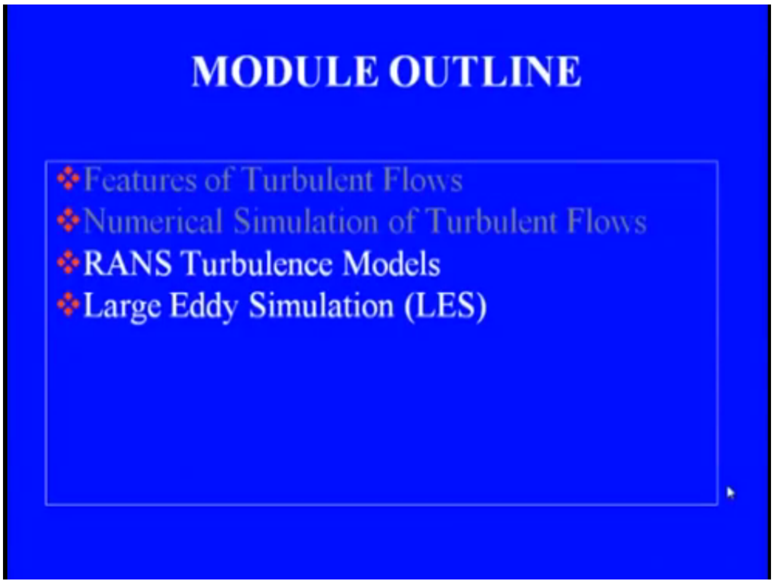
Computational Fluid Dynamics
Dr. Krishna M. Singh
Department of Mechanical and Industrial Engineering
Indian Institute of Technology - Roorkee

Lecture - 41
RANS Turbulence Models and Large Eddy Simulation

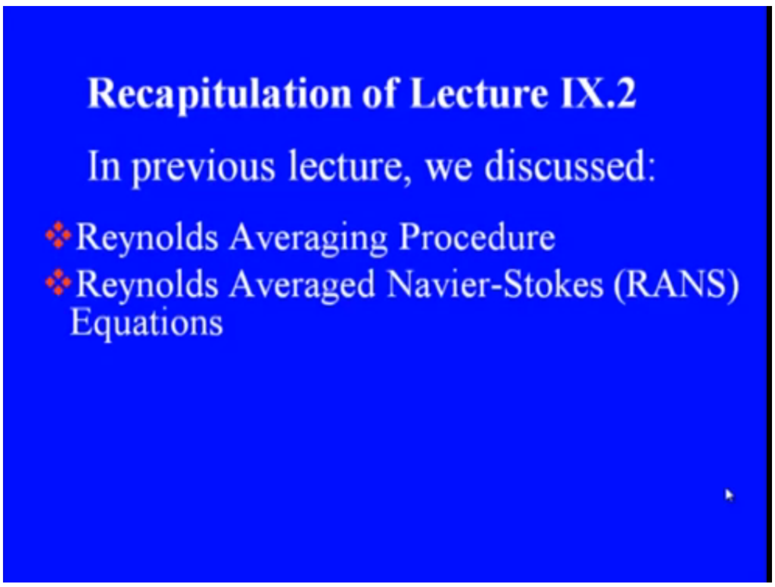
Welcome to the third lecture in module 9 on numerical simulation of turbulent flows.

(Refer Slide Time: 00:31)



So we have already discussed the features of turbulent flows and numerical simulation procedures and we discussed RANS turbulence models in the last lecture and today's lecture we will discuss further on RANS turbulence models and would also discuss large eddy simulation of turbulent flows.

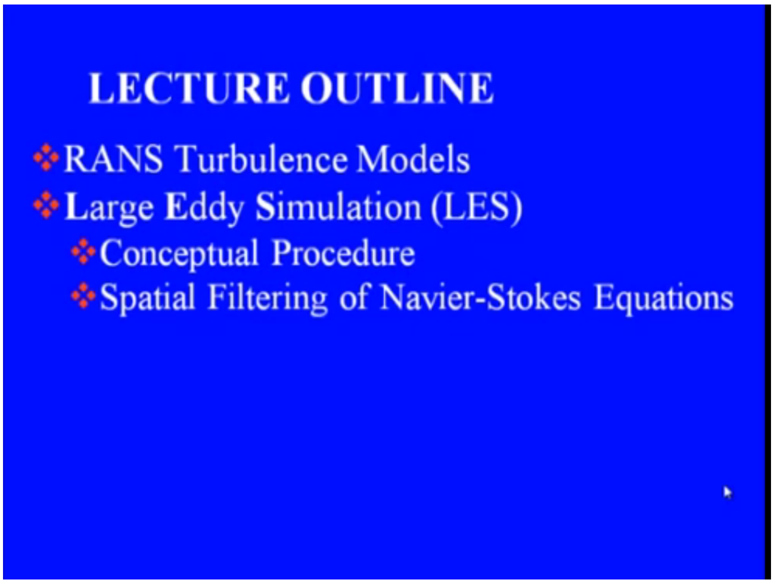
(Refer Slide Time: 00:51)



So let us have a brief recapitulation of lecture 2. We discussed Reynolds averaging procedure and thereby we derived Reynolds averaged Navier-Stokes equations, which contained Reynolds stress terms, which were supposed to be modeled and for that what we need RANS turbulence models. We had discussed a brief outline of Boussinesq proposition and eddy viscosity models.

Today we will have brief look at or rather slightly more detail look at some of the turbulence models used in Reynolds averaged Navier-Stokes simulations.

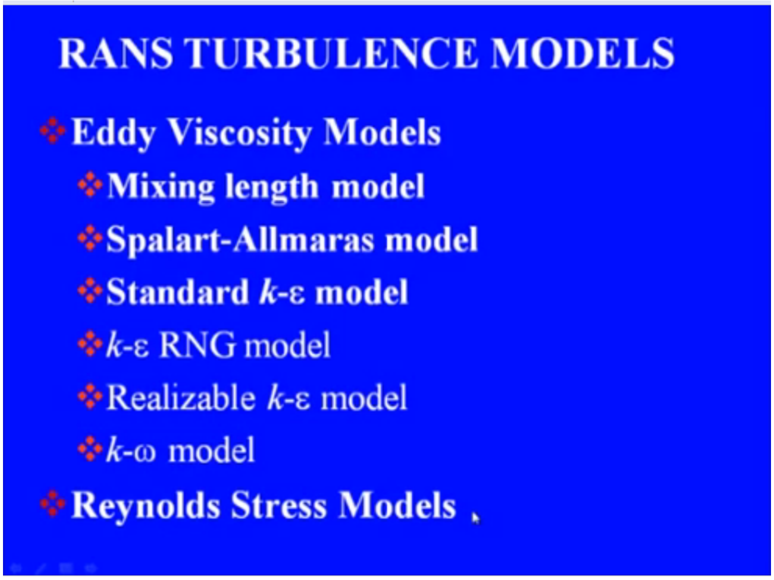
(Refer Slide Time: 01:29)



So focus for today's lecture is RANS turbulence models and then in the second half of the lecture we would consider large eddy simulation. So the outline today's lecture, first we will have a look at RANS turbulence models and then we will look at large eddy simulation, which is called LES in short. We will discuss the conceptual procedure involved in large eddy simulation.

Then we would briefly discuss the spatial filtering operation of Navier-Stokes equations for incompressible flow and then we would have a look at 1 or 2 popular subgrid scale models for large eddy simulation.

(Refer Slide Time: 02:11)

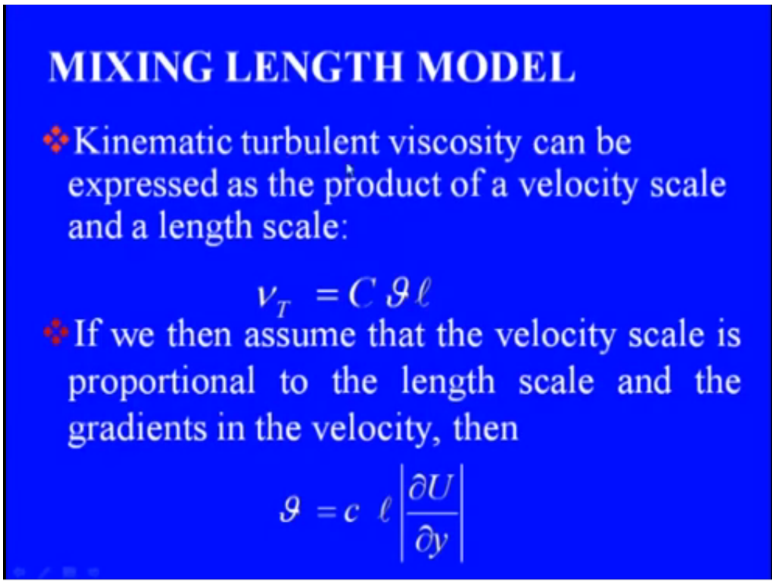


Now in RANS turbulence models, we discussed 2 varieties, one we called eddy viscosity models for instance a mixing length model, Spalart-Allmaras model, standard k-epsilon model, k-epsilon RNG model, realizable k-epsilon model and so on and we also discussed briefly noted briefly the Reynolds stress models. Now in this list with mixing length model is what we call zero equation model.

Spalart-Allmaras model is 1 equation model and standard k-epsilon model is a 2 equation model. Standard k-epsilon, k-epsilon RNG, realizable k-epsilon model or k-omega model, these are all 2 equation models. So we will discuss only one representative model today that is standard k-epsilon model and Reynolds stress model, which directly works on the transport equations of Reynolds equations to obtain the Reynolds stresses directly by solving a set of 6 PDEs.

So the bold ones that is to say mixing length model, Spalart-Allmaras model, standard k-epsilon model and Reynolds stress models for the ones which we are going to have a bit more detail look today.

(Refer Slide Time: 03:25)



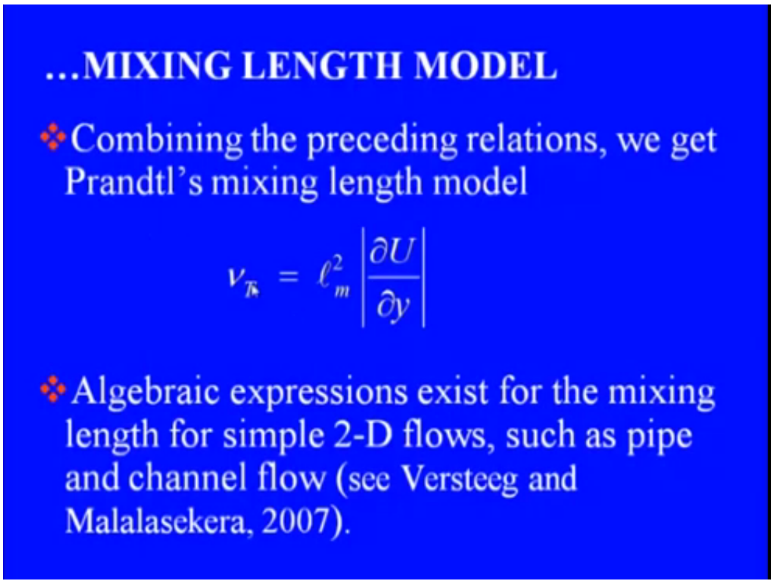
So let us have a look at the mixing length model, which is one of the oldest one which was proposed by Prandtl and the basic premise was based on the realization that look less 2-dimensional analysis and through dimensional analysis we can say that look our kinematic turbulence viscosity, which has got units of meter square per second can be expressed as a product of a velocity scale from where we get meter by second.

And length of scale which will give us m*m/s that gives us m square s and we can multiply it by a constant. So nu tau which our kinematic turbulence viscosity can be expressed in terms of a dimensionless constant and a velocity scale and a length scale. Now this length scale would be typical of our eddy’s structure. Now we can assume that velocity scale is proportional to the length scale and gradients in the velocity field.

This is not consistent with our earlier discussions on turbulent flows that is to say the eddy’s which are there of a particular length their velocity scales are dependent on the gradients of flow velocity. So it is based on this particular observation, which we had a look earlier that we can express the velocity length scale theta in terms of the length scale and the velocity gradient.

Let us say here we are dealing with the pair of flows we can say del u/del y where u is the flow velocity in x direction and y is the length scale or the coordinate perpendicular to that. Now we can combine these 2 equations and thereby we get what is referred to as Prandtl’s mixing length model.

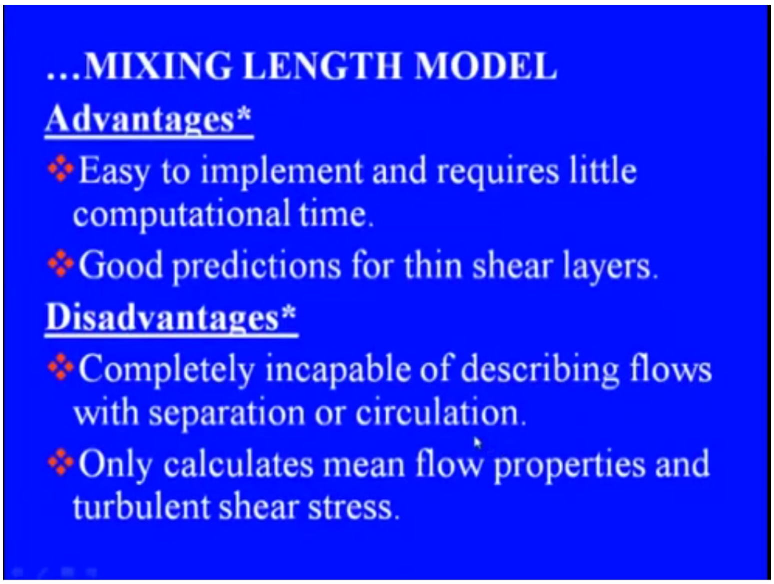
(Refer Slide Time: 05:09)



The name of Prandtl is associated because he was the one who derived it first so nu tau=lm square del u/del y. We have combined the length of scale l and the constants in this lm and for different flows all that we need to know to use this particular model to get an expression or rather a formula, which can give us the value of the mixing length lm, del u/del y that would be obtained from a flow solution itself.

So that is why we do not have to solve a separate equation to compute our eddy viscosity here. Now algebraic equations exist for the mixing length for simple 2-dimensional flows such as pipe and channel flows. For more details, please see the book of Versteeg and Malalasekara.

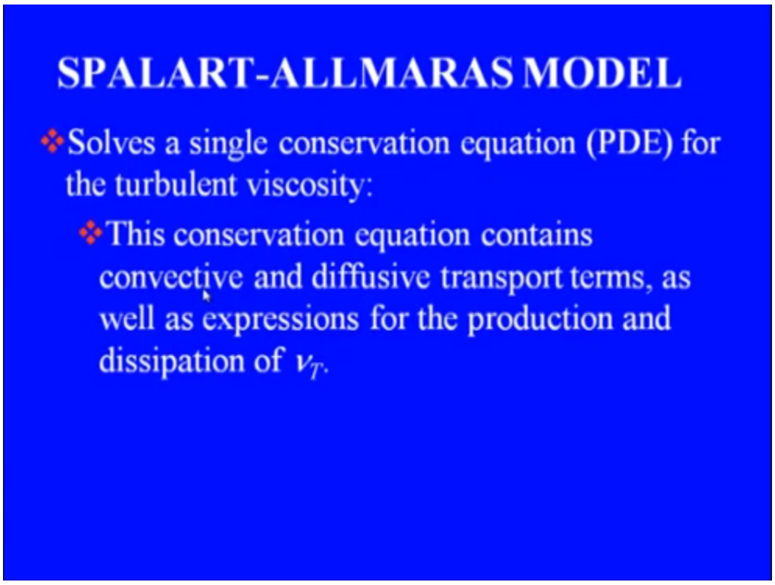
(Refer Slide Time: 06:05)



Now let us have brief look at the advantages of this model. It is very easy to implement and requires very little computational time in addition to solving our velocity components and it gives pretty good prediction for thin shear layers and disadvantages are it is completely in capable of describing flows with separation or circulation and it calculates only mean flow properties and turbulence shear stresses.

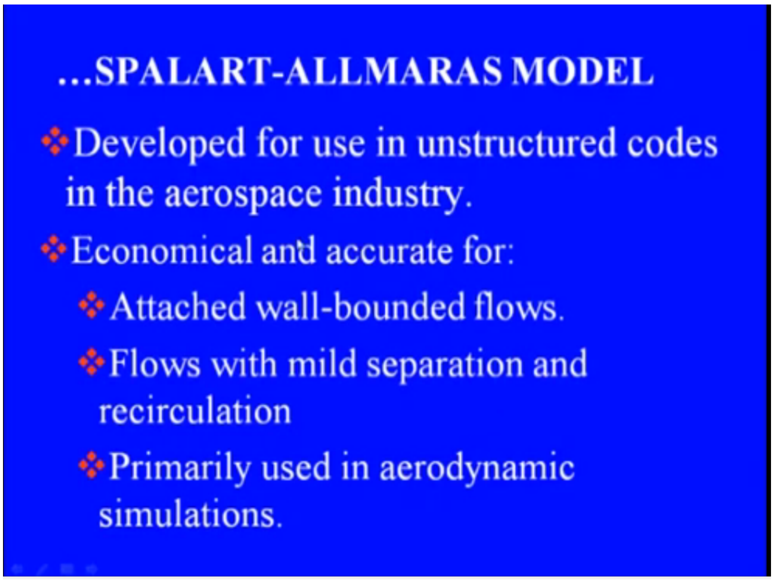
So this mixing length model is used only for the simulation of this shear layers okay. Next category is Spalart-Allmaras model. We will not have a look at the algebraic details because this is very specialized model for aerodynamic calculations.

(Refer Slide Time: 06:35)



And it is a 1 equation model because it is solved for single conservation equation or 1 PDE for the turbulent viscosity and this conservation equation, which has been derived for aerodynamic flows. It contains a convective and diffusive transport term as well as expression for production and distribution of the turbulent kinematic viscosity.

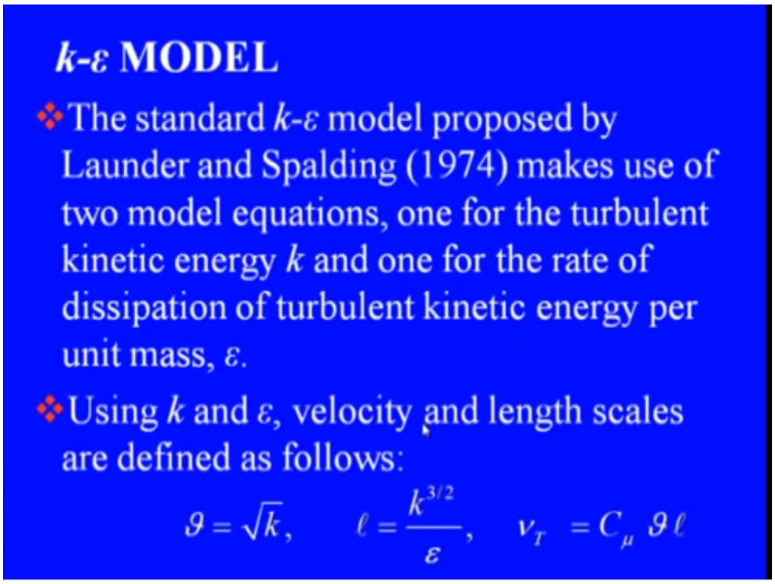
(Refer Slide Time: 06:56)



Now this was used primarily or this was developed primarily for using unstructured codes in aerospace industry and it is economical and accurate for certain type of attached wall-bounded flows and flows with mild separation and recirculation. Please remember the way we described earlier also that it is primarily used in aerodynamic simulation so we are not going to have any detailed look at the equations involved.

If you are interested you can pick up any book references Versteeg and Malalasekara's book or any other CFD book can give you details of this model.

(Refer Slide Time: 07:30)



Next, we come to the most widely used model in fact it is k-epsilon model. It is a 2 equation model, which is the workhorse for industrial CFD analysis. This is one of the most popular models and the so called standard model that the original k-epsilon model it was proposed by

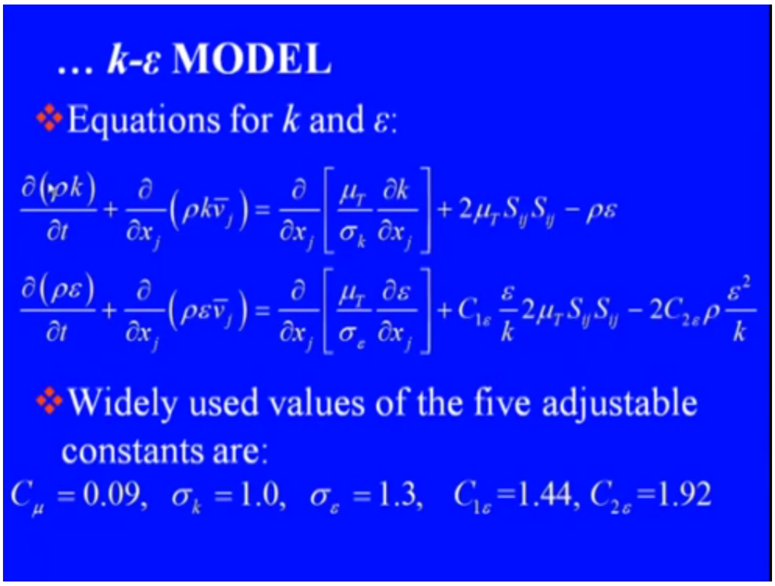
Launder and Brian Spalding in 1974. It makes use of 2 model equations, 1 for turbulent kinetic energy k and one for the rate of dissipation of turbulent kinetic energy per unit mass epsilon.

So this reason we call it as k-epsilon model. So we can solve for these k and epsilon everywhere in the flow field. So we can use this k and epsilon to obtain velocity and length scales. For instance, the velocity scale could be easily obtained by taking the square root of k. Remember k is turbulent kinetic energy per unit mass. So that is basically has got the units of velocities or dimensions of velocity square.

So this square root will give us a velocity scale and length scale is defined in terms of the turbulent kinetic energy that is k to the power 3/2/epsilon so length scale is the one which relates both the kinetic energy of turbulence to the dissipation of the turbulence kinetic energy and we can use these 2, l and theta in our eddy viscosity model to estimate the new tau that turbulent kinematic viscosity as C mu theta*l.

Where C mu is a dimensionless constant. Now this turbulent kinetic energy equation can be derived from our Navier-Stokes equations and invoking Reynolds averaging procedure, but the derivation of equation for epsilon that is bit more complicated, it involves very empirical constants here.

(Refer Slide Time: 09:33)



So we will just list the 2 standard equations and the way they are used. The first one is equation for the kinetic energy k or what we call transport equation for turbulent kinetic energy k. So del/del t of rho k+del/del xj of rho k vj bar=del/del xj of mu T/sigma k del k/del xj+twice of mu T S ij S ij-rho times epsilon. So here epsilon is our dissipation term, this S ij that is velocity rate tensor and strain rate tensor, which can be obtained from the average velocities that is vj bar.

And this mu T is our dynamic eddy viscosity, sigma k is a constant. Similarly next transport equation for epsilon, del/del t of rho epsiolon+del/del xj of rho epsilon vj bar=del/del xj of mu T times sigma epsilon del epsilon/del xj+C1 epsilon epsilon/k 2 mu T S ij S ij-twice of C2 epsilon rho epsilon square/k. So these 2 are coupled partial differential equations. In fact, they also involve the 8 unknown velocity components.

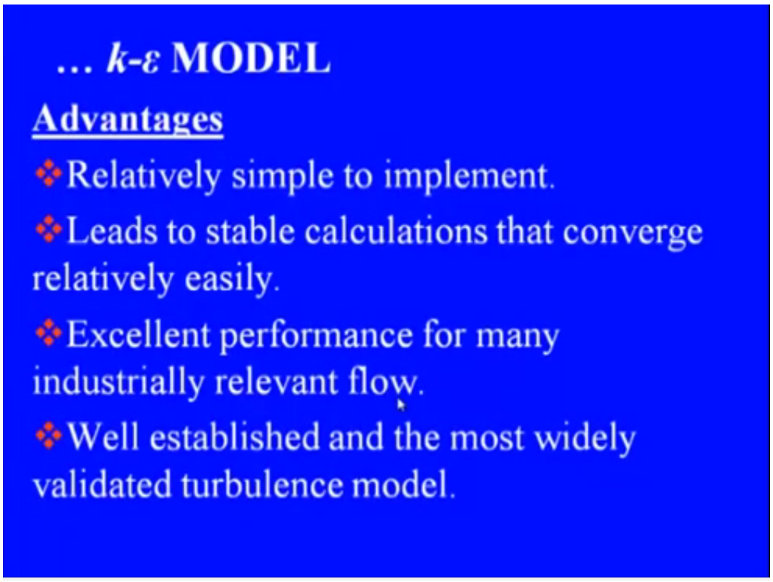
So these two equations for k and epsilon, these two PDEs must be solved together with our 3 momentum equations that is 3 equations for velocity components and we had many empirical constants here sigma k and sigma epsilon, C1 epsilon, C2 epsilon and C mu. So these are adjustable constants, which have been determined or rather they have been estimated based on the curve fitting from a wide collection of experimental data on turbulent flows.

And most widely used values, which have been suggested based on these compilations are that C mu is taken as 0.09, sigma k is taken as 1, sigma epsilon is taken as 1.3, C1 epsilon is 1.44 and C2 epsilon is taken as 1.92. Now these are adjustable constants so depending on the flow situations or flow type you might choose a better value if such an indication is available in literature.

Otherwise will just go by these most widely used values in our code for the solution of k and epsilon and once we know k and epsilon, mu T can be evaluated using the previous formula, which we have seen earlier. Now what are advantages of the k epsilon model? It is relatively simple to implement. We have got 2 additional equations, which have got similar form to that of our normal momentum equations.

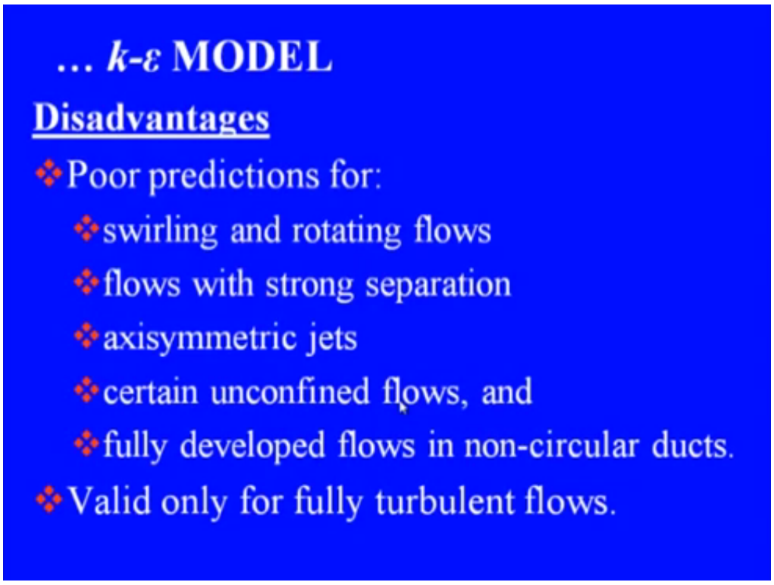
For instance, we had a time derivative term, a convective term, a diffusion term and something like a source term. So solution process would be exactly the same as the one which we are going to use to integrate our momentum equations.

(Refer Slide Time: 12:48)



We have to just add up 2 additional equations and it has been observed that k-epsilon model leads to stable calculations that converge relatively easily, satisfied excellent performance has been observed for many industrially relevant flows and that is one of the reason why this is one of the most widely validated and well established models, which I would say is a workhorse for industrial CFD analysis.

(Refer Slide Time: 13:20)

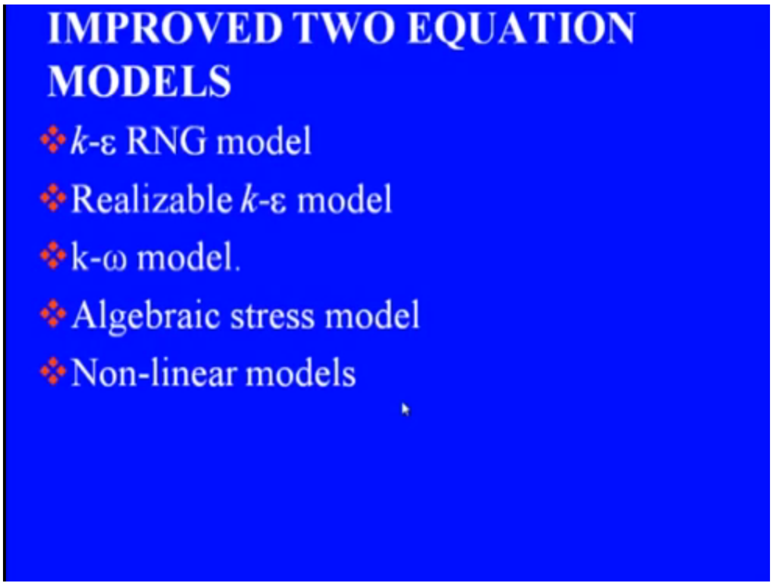


There are some disadvantages though which are linked basically to the way we had assumed a scalar eddy viscosity. So it provides poor prediction swirling and rotating flows wherein we cannot have a scalar kinematic viscosity assumption that is enforced very well. Similarly flows with the strong separation, axisymmetric jets and certain unconfined flows and fully developed flows in non-circular ducts.

Further this model valid only for fully turbulent flows. If you got the low Reynolds number flows, we have got to use special versions of k-epsilon model for low Reynolds number flows (()) (14:01) for the flows which might be just turning into the laminar flows that is transitional flows and a mix of turbulent flows for that we have to use low re versions of k-epsilon model.

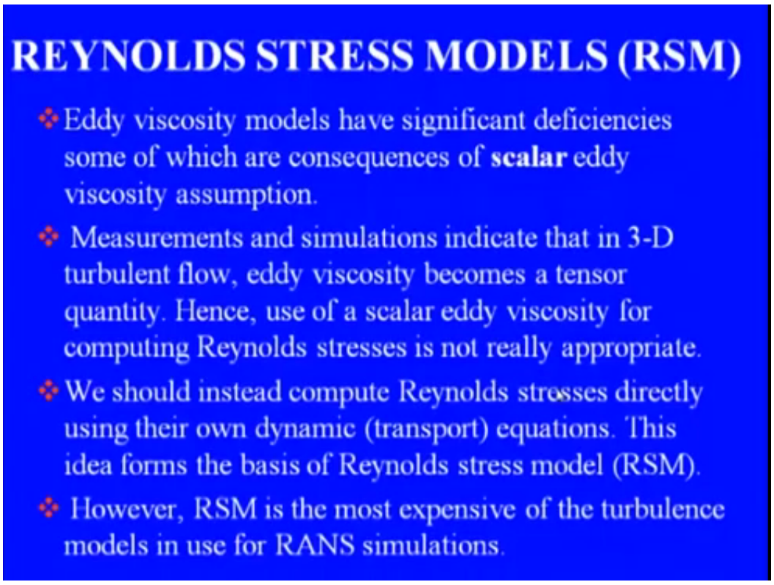
For details, please have a look at the book by Versteeg and Malalasekara on computational fluid dynamics.

(Refer Slide Time: 14:22)



And there are many improved 2 equations models like k-epsilon, RNG model, realizable k-epsilon model, k-omega model and algebraic stress models and there are few nonlinear models as well. We will not discuss in this lecture today. Further details please have a look at the book by Versteeg and Malalasekara. Next let us have a brief look at Reynolds stress model, which is not based on eddy viscosity assumption and why do we need it?

(Refer Slide Time: 14:55)

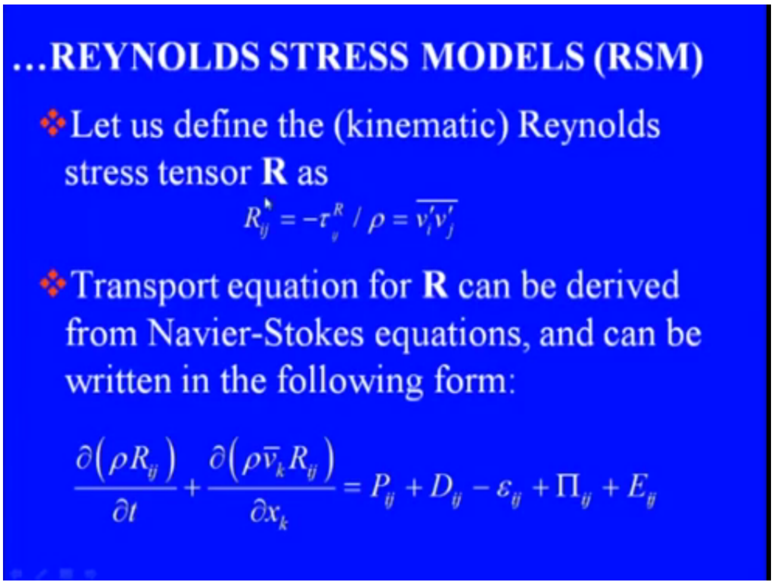


There are some reasons here that this eddy viscosity model they have significant deficiencies, which are consequences of assumption of scalar eddy viscosity and experimental measurements and direct numerical simulations indicate that in 3 turbulent flows specifically over the complex geometries, eddy viscosity is not a simple scalar quantity in fact it has got directional preferences so it becomes a tensor quantity.

Hence, the use of scalar eddy viscosity for computing Reynolds stress is not really appropriate and that is the reason why k-epsilon model does not work very well for such situations. So we should instead now try to compute Reynolds stress directly using their own dynamic or transport equations. So this particular observation is what forms the basis of Reynolds stress model that is to say let us derive the transport equation for Reynolds stresses and then try to solve them.

But it is not as simple as it appears to be. First is one of the most expensive of turbulence models and nevertheless though we can derive transport equation for Reynolds stresses starting from Navier-Stokes equations and Reynolds averaging procedure, we will quickly realize there are more terms in which new terms appear, which need to be modeled.

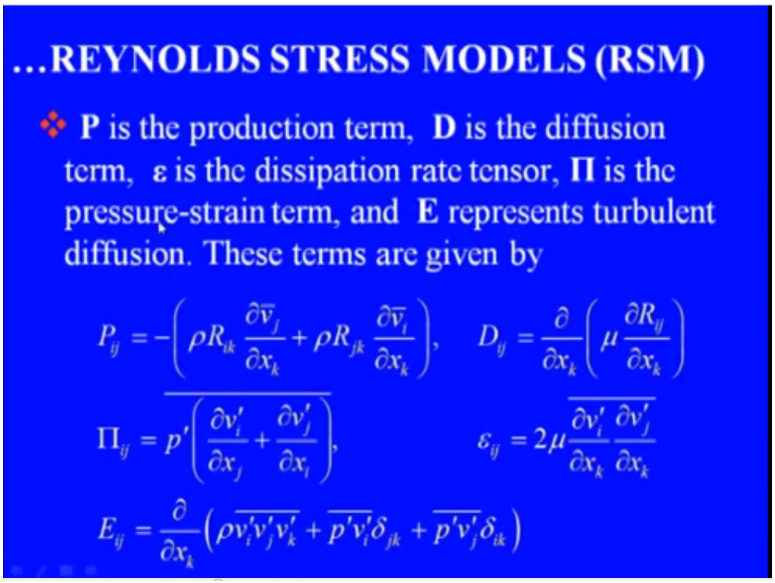
(Refer Slide Time: 16:10)



So now let us define the kinematic Reynolds stress tensor capital R as R ij=-tau ij R/sigma that is it is simply equal to vi prime vj prime overbar. So this is the average of flux product of fluctuating components vi prime and vj prime. Now transport equation for R can be derived from Navier-Stokes equations and we can write it in following form. The first term is the time varying term, the second one is sort of a convective term for a transport of Reynolds stresses.

And the right hand side we have got few additional terms there details we will have a look at in the next slide. So let us go how the equations reads like del of del t of rho R ij+del of del x k of rho vk R ij=P ij+D ij-epsilon ij+pi ij+E ij.

(Refer Slide Time: 17:19)



Now here P is production term, D is called diffusion term, epsilon dissipation rate tensor, capital pi is pressure-strain term and E represents turbulent diffusion. So P ij is given by –(rho

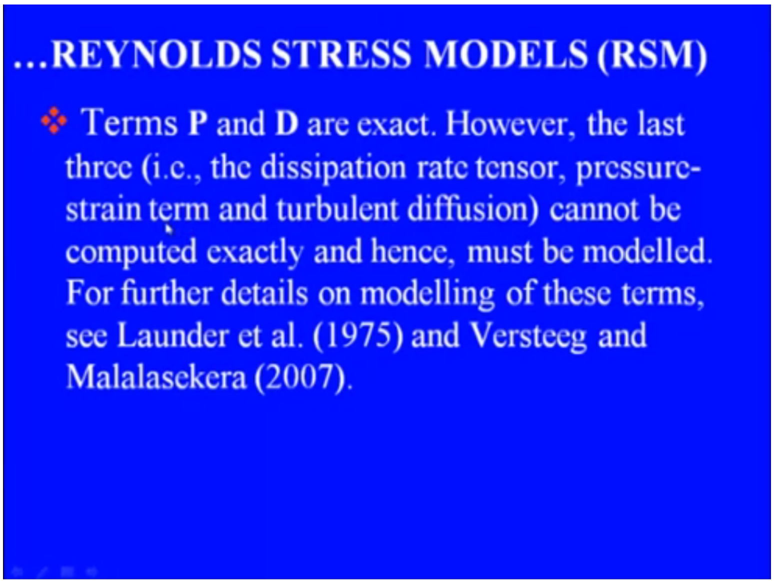
R ik del vj/del xk+rho R jk del vi/del xk. So if you look carefully this involves only our Reynolds stress terms and Reynolds average velocity components so this can be easily calculated.

The next our diffusion term D ij is again del/del xk of mu del R ij/del xk this is the second relevant term, which does not involve any further unknowns but let us have a look at pi ij. This is an average of the fluctuations of pressure and the fluctuating strain rate that is p prime, del vi prime/del xj+del vj prime/del xi time average of all of this. This is something which we do not know. So this has to be modeled.

Similarly, for epsilon ij which is dissipation rate tensor is 2 mu del vi prime/del xk*del vj prime/del xk overbar that is also got to be computed. This also involves unknown correlations. The same words for E ij which is del/del xk of rho vi prime vj prime vk prime overbar, which is what we call a third order moment+p prime vi prime delta ijk overbar+p prime vj prime overbar delta ik.

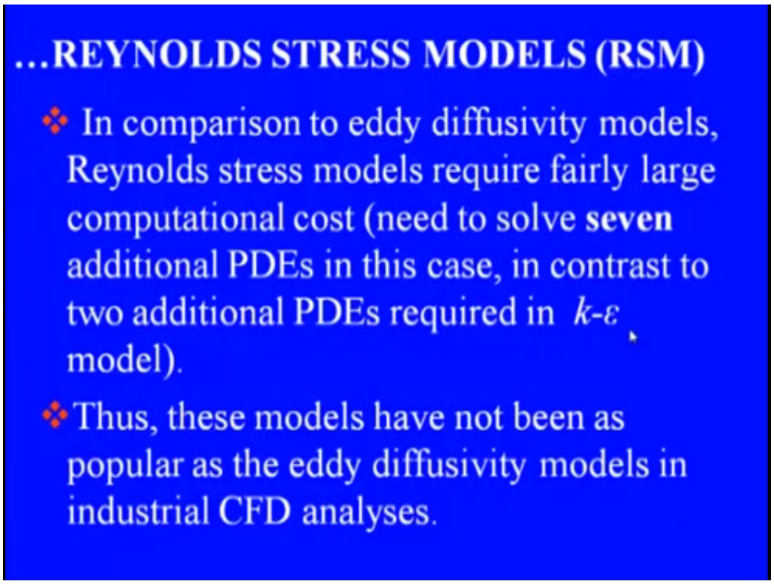
So now we have introduced additional unknowns through these terms and these ones like capital P ij, epsilon ij and E ij they must be modeled okay.

(Refer Slide Time: 19:18)



So that is what is summary here that these ones the dissipation rate tensor, pressure-strain term and turbulent diffusion term they cannot be computed exactly and hence must be modeled. There were many different models, which have been proposed by different researches, which are available in literature. So for further details on modeling of these terms please see the first initial paper of Launder et al in 1975 and the book by Versteeg and Malalasekara, which gives you further references regarding many different proposed models for modeling these 3 terms.

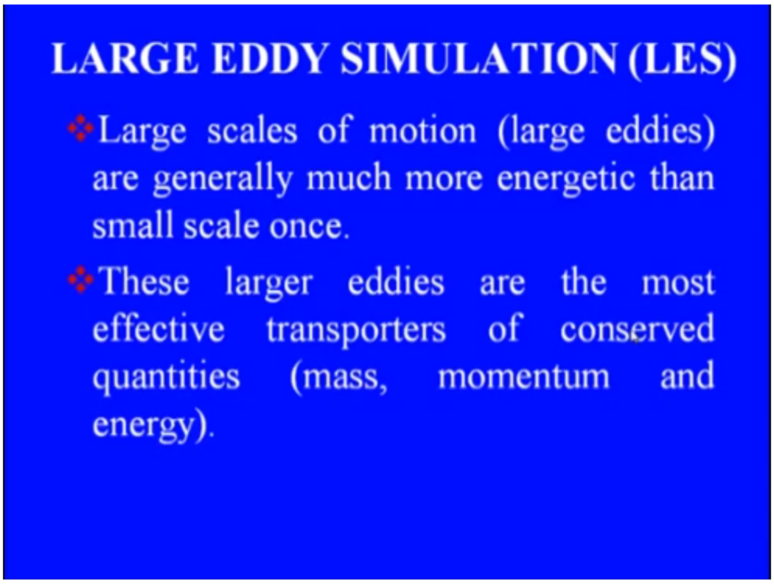
(Refer Slide Time: 19:50)



So that is why we would put a full stop to our discussions on Reynolds stress models with just one observation that in comparison to eddy diffusivity model, Reynolds stress models require fairly large computational cost that is very obvious because we need to solve 7 additional PDEs, 6 for Reynolds stresses and 1 for dissipation term in contrast 2 required in k-epsilon model.

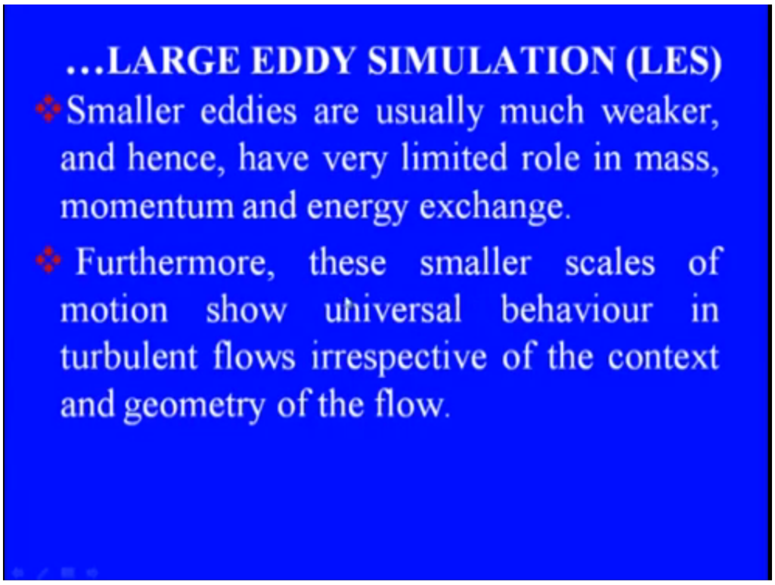
So it is pretty expensive and thus one of reasons why these models have not been as popular in industrial CFD analysis. Next let us move on to our large eddy simulation, which is not picking up in industrial CFD analysis. So what is the basis? Let us have a look at what we have learnt earlier and discussions on turbulent flows.

(Refer Slide Time: 20:30)



And what we have observed that large scales motion, which we call large eddies, they are much more energetic than small scale eddies and what is the consequence? It is the larger eddies which is our most effective transporters of the conserved quantities that is they are the ones who transport mass, momentum and energy. They are largely anisotropic and these are the ones which we need to resolve accurately in our flow simulations.

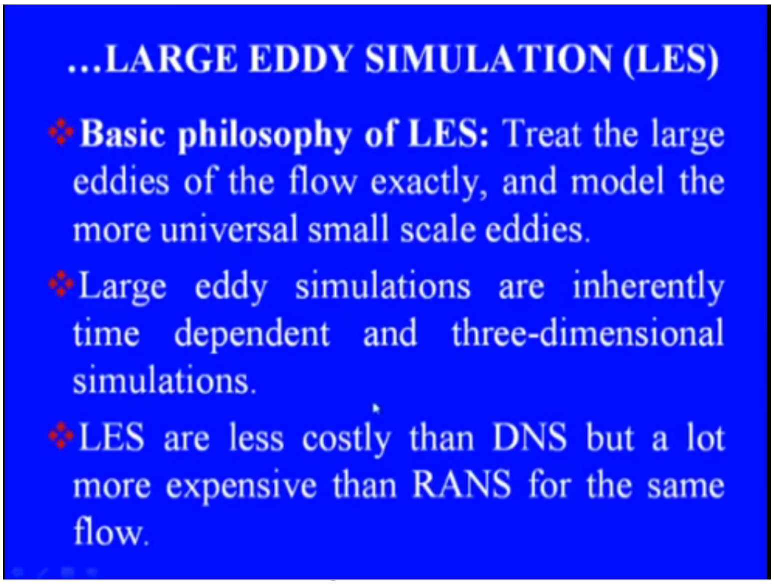
(Refer Slide Time: 21:02)



In contrast to large scale eddies that what we call large eddies, the smaller eddies are much weaker and they have got very limited role to play in the transport processes of mass, momentum or energy. Furthermore, we have seen that small scale motions they show universal behavior. They are more or less isotropic irrespective of the turbulent flow with the turbulent flows takes place in one type of geometry or another type of geometry.

So their behavior is what we call universal irrespective the context Reynolds number or the geometry of the flow. So what we can do is we can instead come up with a model for this smaller eddies. It is lot easier to capture the effect of this small scale eddies through a model and try to resolve the large eddies accurately. So that forms what we call the basis or basic philosophy of LES.

(Refer Slide Time: 22:01)



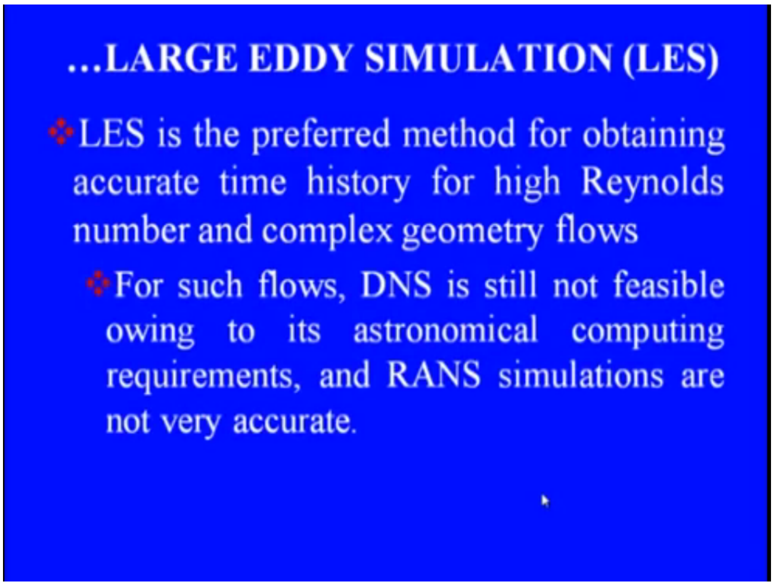
That is to say treat the large eddies of the flow exactly and model more universal small scale eddies through what we would call is subgrid scale model and please remember this large eddy simulations that they are inherently time dependent in 3-dimensional simulations in contrast to statistically averaged or time averaged Reynolds averaged equations where we get steady state flows simulations.

Large eddy simulations are time dependents ones and as a consequence large eddy simulation they are more costly than RANS simulations, but they are much less costly than DNS because in DNS we require to resolve all the scales small scales as well as large scales, our grid size and time steps were dictated by the smallest eddies that is the ones which have a size of Kolmogorov length scale, but in LES we are not going to resolve up to that scale we will rather stop in the middle.

We will just resolve the large eddies, so we can have much larger grid size in large eddy simulation and consequently a large eddy simulation would be much less costly than DNS by maybe few orders of magnitude and at the same time since it is a time dependent 3-

dimensional simulation it could be a lot more expensive than Reynolds averaged Navier-Stokes simulation for the same flow.

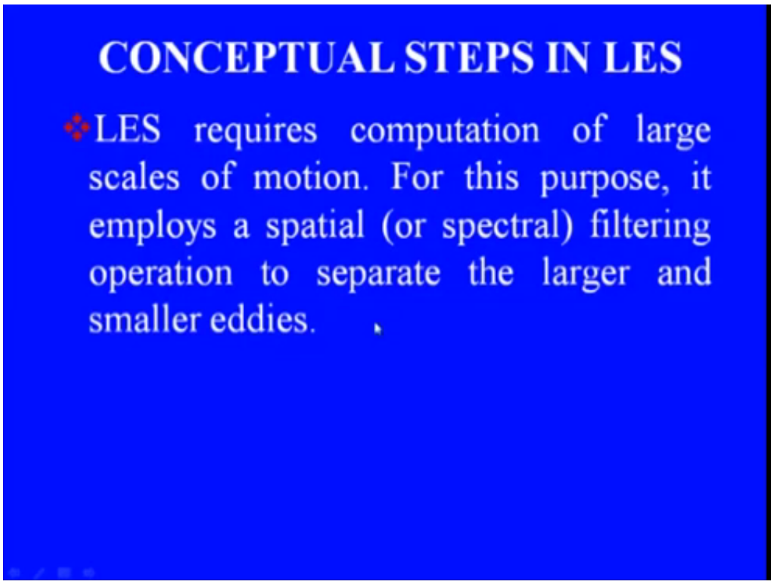
(Refer Slide Time: 23:22)



Nevertheless, LES is preferred method for obtaining accurate time history for high Reynolds number and complex geometry flows for which DNS is not feasible owing to astronomical computing requirements and RANS simulations are not very accurate. We have already seen the k-epsilon model is not very good for the complex geometries flow, which will involve lots of (()) (23:44) and flow separations and so on.

Since such situations large eddy simulation is the one which can help us out. So let us have a brief look at the conceptual steps in large eddy simulation.

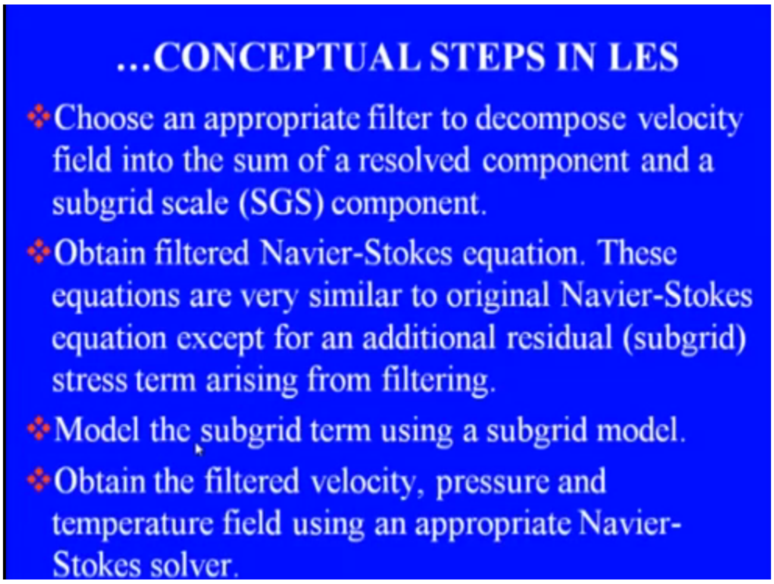
(Refer Slide Time: 23:56)



The main concept involved is that we have to somehow separate the larger scales with smaller ones and we would compute only the large scales of motions. So for this purpose we have to employ a spatial filtering operation to separate the larger and smaller eddies. In case we are working in spectral domain then we have to come up with the spectral filter and thereby separate eddies of high wave number and small wave number.

And once we have separated the filtered or resolved scale field is the one which is simulated that is one which we would compute numerically using a suitable Navier-Stokes solver, which already discussed earlier.

(Refer Slide Time: 24:40)



So now let us summarize the steps involved in large eddy simulation. The first step is choose an appropriate filter to decompose velocity field into sum of a resolved component that is our large eddy component and a subgrid scale are what we call in short SGS component. So this SGS component is what refers to small scale eddies and once we have chosen the filter, we would use that filter to obtain filtered Navier-Stokes equations.

Now these equations would be very similar to the original time dependent Navier-Stokes equation except for an additional residual term, which will call a subgrid stress term arising from filtering. The word stress here is used to just maintain a new formatting with historical perception of the fluctuations length, which we have seen earlier in Reynolds averaged Navier-Stokes simulations.

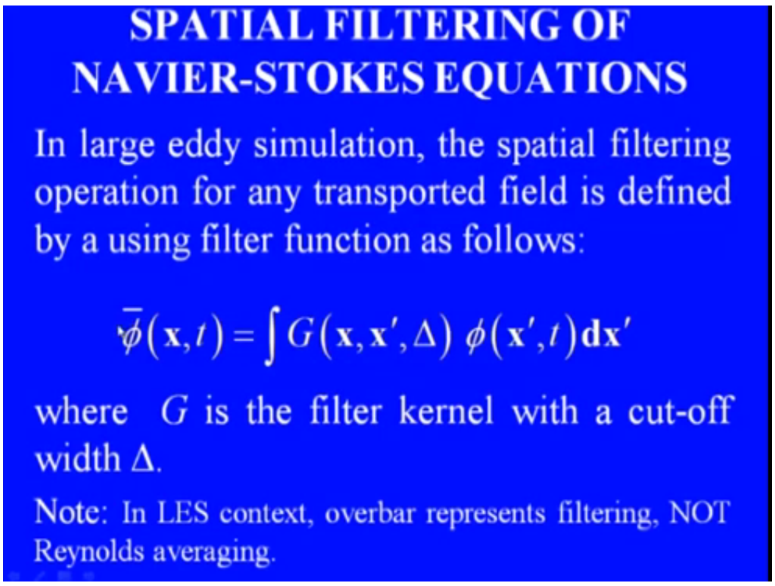
So just keeping that terminology we have used here subgrid stress term and once we have obtained the filter Navier-Stokes equations, the next step would be that this equation will not be close because there would be some additional term that is residual or subgrid system. Now this stress term has to be modeled so that it is expressed in terms of unknown quantities in terms of our resolved velocity field.

So we have to come up with or use a model, which we call subgrid scale model and once that is ready thereafter we can use our numerical simulation procedure, which we have learnt earlier for Navier-Stokes equations to obtain the filtered velocity, pressure and temperature field using an appropriate Navier-Stokes solver. Now remember here it is just more comments in order that large eddy simulation again we would like to have an accurate time history.

So very often in practice what do we do is we choose an explicit Navier-Stokes solver because our time step is given by the accuracy requirements that would usually be pretty small to satisfy the stability requirements imposed by an explicit time integration schemes and since the integration scheme is explicit, time step which is small enough to satisfy the stability requirements.

We can calculate very efficiently our velocity components using this explicit time integration scheme.

(Refer Slide Time: 27:16)

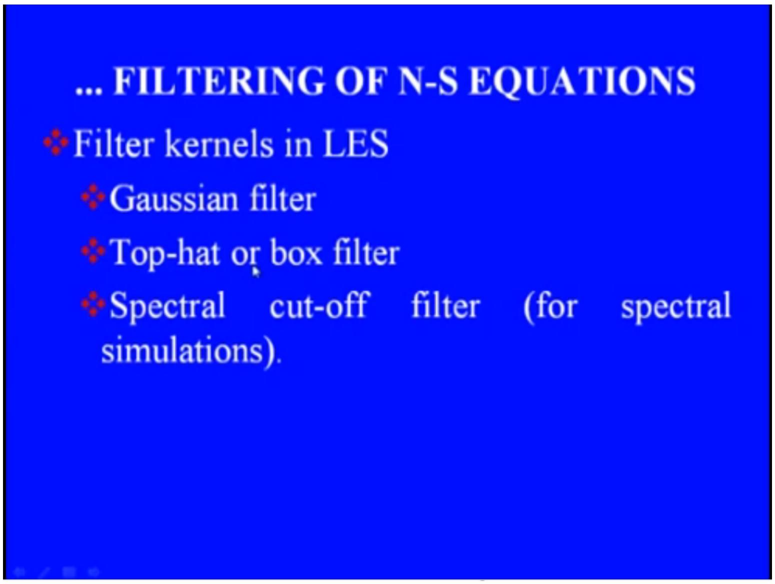


Now let us have a look at the filtering operation in Navier-Stokes equation. For the time being, let us have a look at spatial filtering. The spectral filtering works in analogous fashion okay. So in large eddy simulation, the spatial filtering for any transported field, the transported field could be one of the velocity components, it could be temperature and so on. So it is defined using a filter function.

So suppose phi were our variable or flow variable so phi bar x t is defined as the integral of this convolution G x x prime delta, this is our filter function multiplied by our unfiltered or original flow variable phi x prime t dx prime. So this integration is carried out over the spatial domain and here where G is the filter kernel and this delta is the cut-off width. Now we would see that we will actually do not have to perform a complicated 3-dimensional integrations.

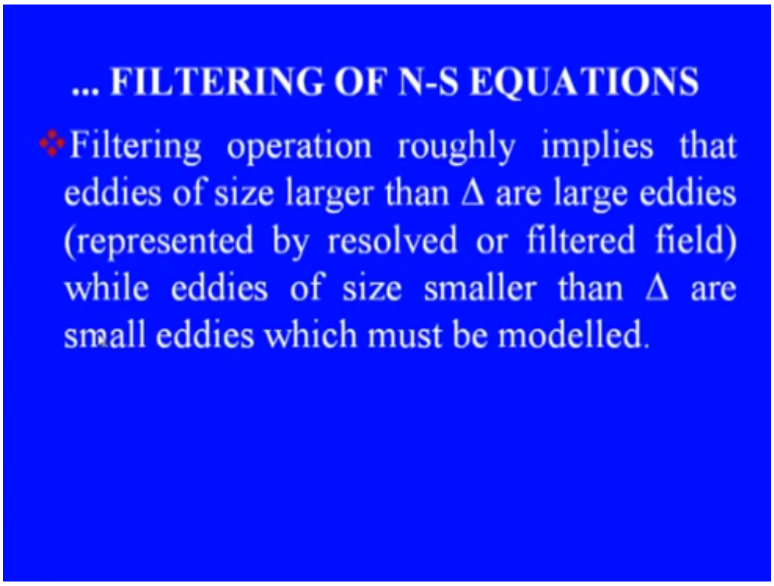
It is just a conceptual step okay and there is one more comment in order. Here we have used overbar, but please remember in the context of large eddy simulation overbar would not denote Reynolds averaging, it denotes a filtered quantity that is to say it denotes a result of applying this filter G x x prime delta on our flow variable phi. So hope there is no further confusion.

(Refer Slide Time: 29:12)



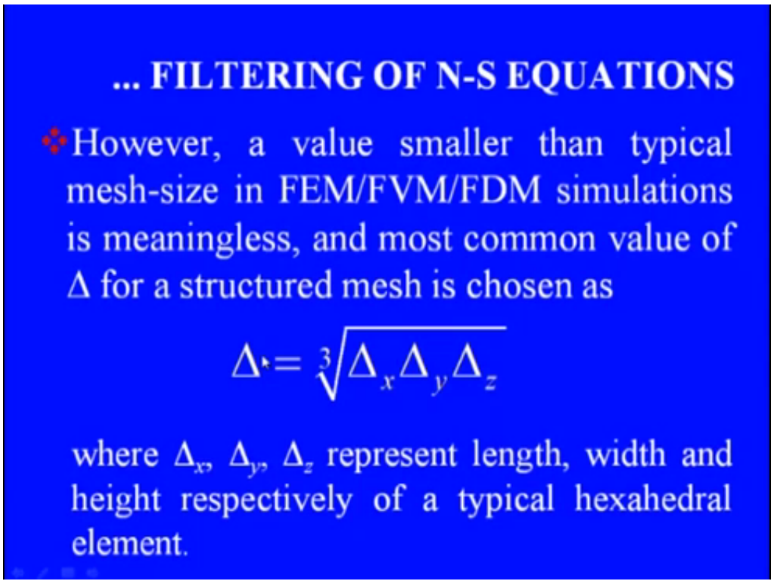
We are going to use this overbar to represent a filtered flow variable. Now some of the popular filter kernels in large eddy simulation is one Gaussian filter, top-hat or box filter which essentially is linked to let us say our mesh size and if you are working for spectral simulation that is to say our numerical simulations are being performed in spectral domain, we can also use spectral cut-off filter.

(Refer Slide Time: 29:24)



Okay let us see what this filtering operation employ? It basically a size that look the eddies of size which are larger than delta they are large eddies and they represent our resolved or filtered field, while eddies of the size smaller than delta are small eddies which are filtered out by this filter and which must be modeled. Now this cut off width delta can in principle have any size back in theory or in practice what do we do?

(Refer Slide Time: 29:55)

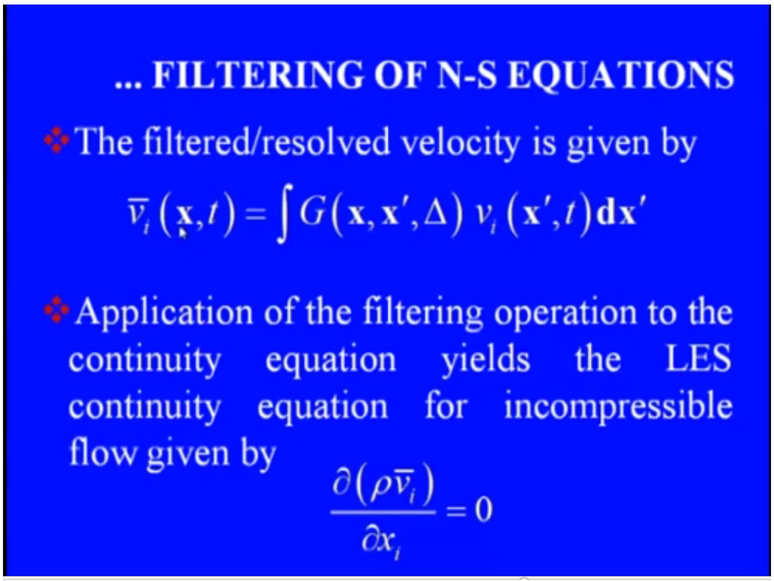


We choose a value which is almost of the same order as that of a full grid size because a value smaller than typical mesh size in our finite element, finite volume or finite difference simulations is meaningless. So the most common value is an averaged mesh size for instance this delta for a structured mesh can be obtained as delta=cube root of delta x delta y delta z where delta x, delta y and delta z they represent length, width and height of a typical hexahedral element.

So you can think of this delta x, delta y and delta z to be the mesh sizes in x, y and z directions on a structured grid in a finite difference method or in finite volume method or they could be sizes of the finite element if you have used hexahedral elements in finite element simulation. Similar estimates could also be obtained for tetrahedral elements in finite element simulations and so on.

So in a nutshell this delta represents an average value or average length linked to our finite element or finite volume cell or finite difference grid.

(Refer Slide Time: 31:04)

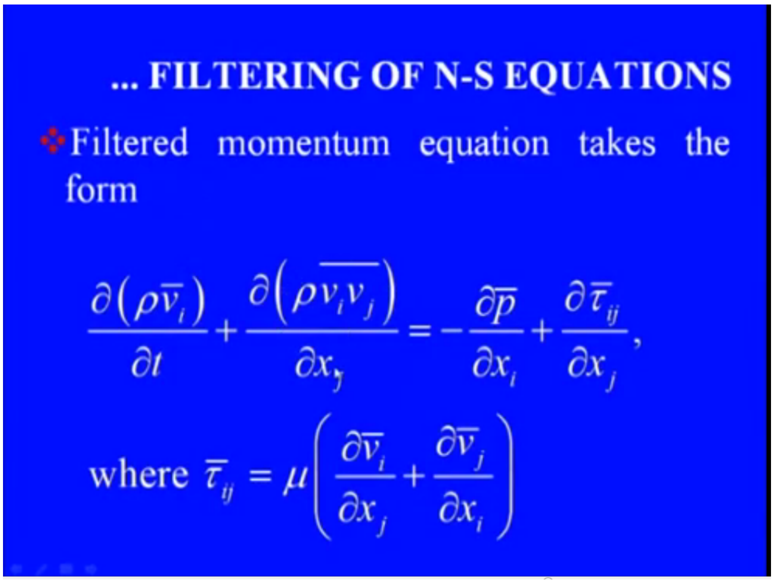


And based on this filtering, filtered or resolved velocity that can be expressed as vi bar x t remember the filtered velocity or any filtered variable for that matter it would be function of a spatial coordinate as well as the time. So this symbolically represents G x, x prime, delta vi x prime t dx prime and application of this filtering procedure is very simple that application of filtering operation to continuity equation for incompressible flows that leads to a very simple form.

Only the time derivative term which is not there for incompressible flow we have got only one term here that is divergence term so del/del xi of rho vi bar=0. So if you remember our original equation was del/del xi of rho vi that was our original continuity equation. The only thing which has changed for the filtered equation is we have just replaced vi by the filtered velocity vi bar.

Similarly let us now apply the filtering process to our momentum equation. So we can substitute each term or each variable term in our momentum equation by the filtered counterpart and that would give us the filter momentum equation for instance.

(Refer Slide Time: 32:26)



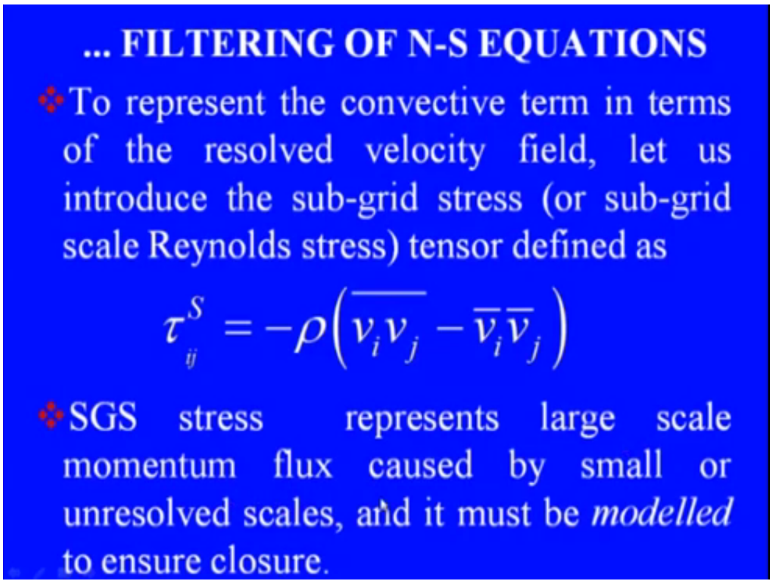
And the second term is the convective term, the third term is the pressure gradient and the last one is our stress term okay. So if we remember our original equation was simply del/del t of rho vi+del/del xj of rho vi vj=-del p over del xi+del tau ij/del xj. I have noticed the source term because that would remain unchanged from the filtering operations. If it is required we can always incorporate here.

In majority of the flows the source time can be observed in a pressure term so that is why we have not put here explicitly. So remember what we have done. What is the net effect of filtering operation? Vi has been replaced by vi bar in the first term so the temporal term becomes del/del t of rho vi bar. Similarly in the second term the convective term, vi vj bar this product is replaced by vi vj overbar.

So this convective term becomes del/del xj of rho vi vj overbar=-del p bar/del xi. We have just replaced p by the filtered pressure field+del/del xj of tau ij bar. Now tau has been replaced by the filtered counterpart. So this tau ij is defined as mu times del vi bar/del xj+del vj bar/del xi that is to say all that we have done is in this definition of tau ij we have just replaced a vi and vj by vi bar and vj bar that is the filtered velocity components.

So this equation has got a form very similar to our original Navier-Stokes equation except for this convective term because in convective term now we have got here a cross correlation term vi vj overbar. So we have now somehow got to change this term and for that purpose what do we do?

(Refer Slide Time: 34:36)

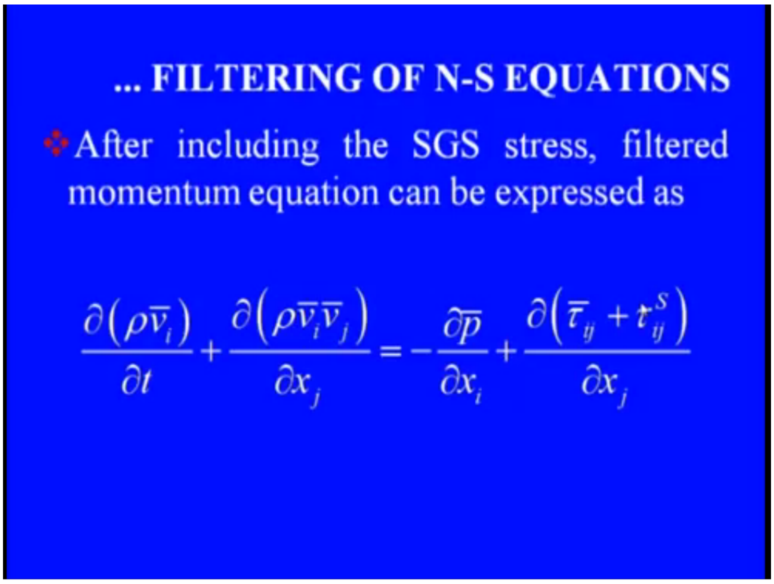


To represent the convective term in terms of the resolved velocity field let us introduce subgrid stress term and this is also referred to as subgrid scale Reynolds stress term in analogy with our Reynolds stress models or what we call Reynolds averaged Navier-Stokes equation where we call a term involving velocity fluctuations as a Reynolds stress term. In LES literature, some people prefer to use a term subgrid scale Reynolds stress term.

And this particular tensor is defined as tau ij let us use a super script S to differentiate it from our Reynolds stress and RANS models. So tau ij super script S=-rho (vi vj overbar that is the filtered value of this product of vi and vj-vi overbar into vj overbar) and introduction of this would be now this –rho times vi overbar vj overbar that can be transferred to the left hand side and the remaining terms could be observed put in the right hand side to get a form which is now analogous to our original Navier-Stokes equation.

Now this subgrid stress what does it represent? It represents the large scale momentum flux which is caused by small or unresolved scales and since it involves a term which we cannot compute explicitly using our filtered velocity values that is vi bar and vj bar, this term vi and vj overbar cannot be computed in terms of vi bar and vj bar explicitly. So this whole term this tau ij S it has to be modeled to ensure the closure of filtered Navier-Stokes equation.

(Refer Slide Time: 36:38)

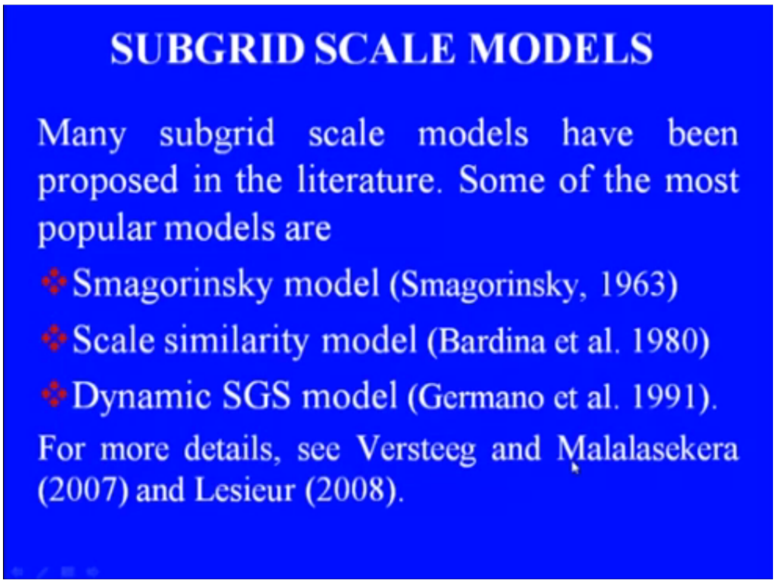


And this modeling is referred to as subgrid scale modeling. Now let substitute this tau ij S in our filtered equation and rearrange the terms del rho vi bar/del t+the next one is convective transport term del/del xj of rho vi bar vj bar=-del p bar/del xi so this represents a pressure gradient term+del/del xj of tau ij bar+tau ij S. Now tau ij bar that is represented in or expressible in terms of velocity strain rate tensor.

And this tau ij S that is something which we have to obtain using our subgrid scale model, but if you look at the form this form is exactly same as our original Navier-Stokes equation except for this additional term on the right hand side, which we would compute somehow and incorporate. So any code which we have written to solve our Navier-Stokes equation so any standard Navier-Stokes solver can state may be used for large eddy simulation.

We do not have to solve for separate partial differential equation in this case the way we did in the case of the RANS models typically let us say our k-epsilon model or Reynolds stress model where we have to solve for additional partial differential equation, here we do not require that in our large eddy simulation. Now we have to find out the expression for this subgrid stress term.

(Refer Slide Time: 38:07)

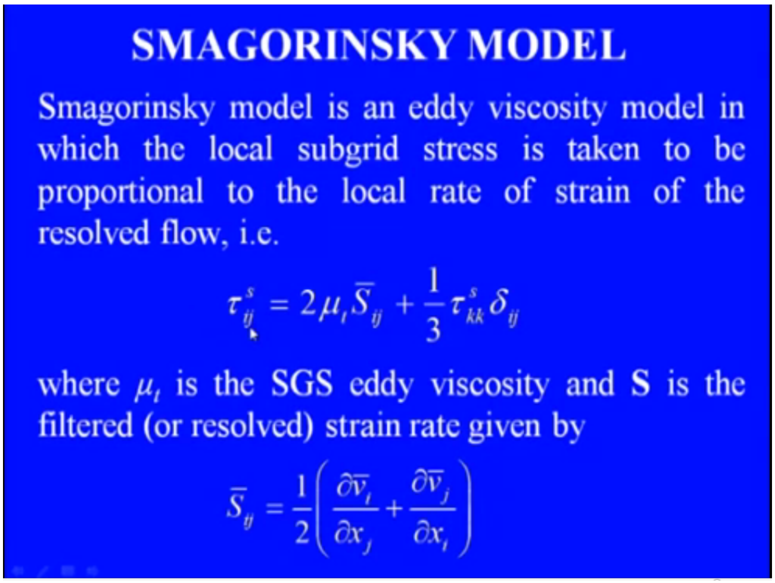


So there are many subgrid scale models have been proposed in the literature starting from Smogorinsky model in 1963. Smogorinsky was the first person to come up with the large eddy simulation for atmospheric boundary layer and there he proposed a very simple model in analogy with the Boussinesq eddy viscosity proposition and there are many improvements proposed over Smogorinsky model.

So one is called scale similarity model, which was proposed by Bardina et al in 1980. Then dynamic subgrid scale model. I have just mentioned only the most popular ones of them here. There is famous research going on to come up with better and better subgrid models based on the DNS data and very accurate experimental results. So for more details, please see the book on computational fluid dynamics by Versteeg and Malalasekara.

And for even more details you can see the book on turbulence by Lesicur in 2008. In fact, this one full book by Lesicur, which is devoted to large eddy simulation. So that will contain the models developed up to year 2004 and still research work continues to obtain more and more improved large eddy simulation models.

(Refer Slide Time: 39:50)

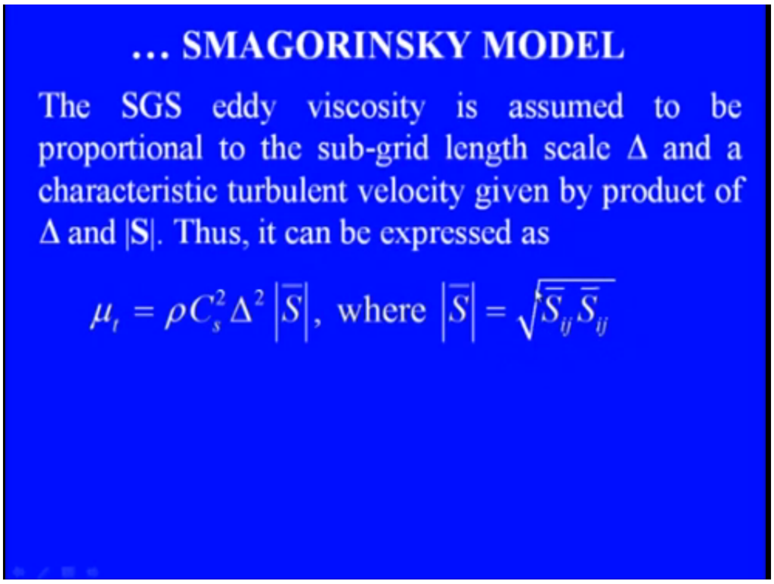


Now let us have a look at Smogorinsky model. So Smogorinsky model is an eddy viscosity model that is how its inspiration came from there from the eddy viscosity model, which we have discussed earlier in the context of Reynolds stress or Reynolds averaging process. So in this case, the local subgrid stress is taken to be proportional to the local rate of strain of the resolved flow.

That is to say this tau ij S is our subgrid stress tensor=2 mu t purposely use small t here to say that this mu subscript small t is our subgrid scale eddy viscosity. So tau ij S is twice of mu t S ij bar which S ij bar is our strain rate tensor based on resolved velocity field that is S ij is half of del vi bar/del xj+del vj bar/del xi okay+next term is 1/3 tau kk s delta ij. So this is Smogorinsky model.

Now here to use this model we need an estimate in the modeling constraint how do we obtain an estimate of mu t? So that is the modeling part okay. So what Smogorinsky proposed is this SGS eddy viscosity it is proportional to the subgrid length scale delta and a characteristic turbulent velocity.

(Refer Slide Time: 41:08)

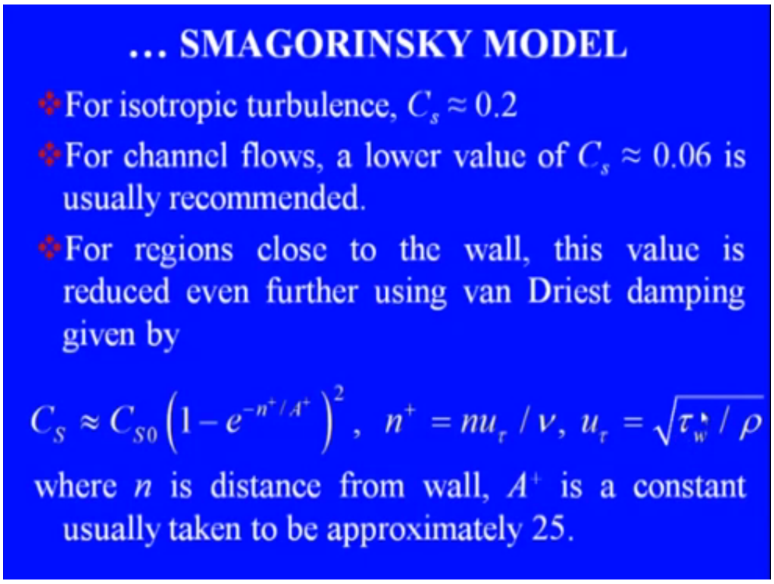


Now a turbulent velocity can be obtained by the product of the length scale delta and a scalar product which we can obtain from our strain rate tensor. So let us call it as magnitude of S so this we can write mu t S rho times Cs square delta square S bar where this S bar magnitude is given as square root of S ij overbar S ij overbar. So this S ij overbar*S ij overbar this is a doubly contracted tensor product of a strain rate tensor.

So that will give us a scalar quantity. Take the square root of that that is what this particular symbol S bar modulus represents. So this S bar can be easily evaluated from our resolved velocity field, we can easily calculate that, delta is known to us, rho is of course known to us and CS is a constant value, which we call a model parameter. Now this CS is not a universal constant.

It depends on the type of flow and different values have been suggested for different types of flows.

(Refer Slide Time: 42:43)

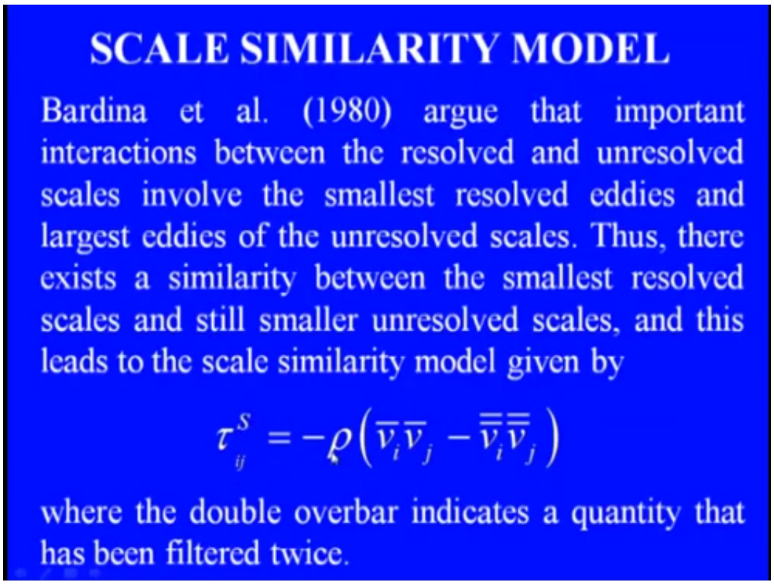


For instance, for isotropic turbulence CS was taken as approximately 0.2, for channel flows a lower value of CS=0.06 or some people use 0.1 that is usually recommended. Similarly for wall-bounded flows if you use this when CS=0.06 or 0.1 is too larger value. So yet another modification was suggested that for reasons close to the wall this value is reduced further using van Driest damping.

This was proposed by van Driest and that is why it is called as van Driest damping that is to say reduce the value of this constant or this dimensionless number any valuation for eddy viscosity further. So CS close to the wall would be expressed as CS 0 where CS 0 could be 0.1 or 0.06 (1-exponential of –n+/A+) whole square where this n+ represents a non-dimensional distance involved that is n+=n u tau/nu.

Nu is our kinematic viscosity, u tau is what we call wall shear velocity, which is given in terms of shear stress of the wall that is u tau=square root of tau W/rho and A+ is a constant whose value is approximately taken as 25. So this is our van Driest damping and large eddy simulation based on Smogorinsky model normally makes use of the van Driest damping close to the wall for wall-bounded flows.

(Refer Slide Time: 44:19)

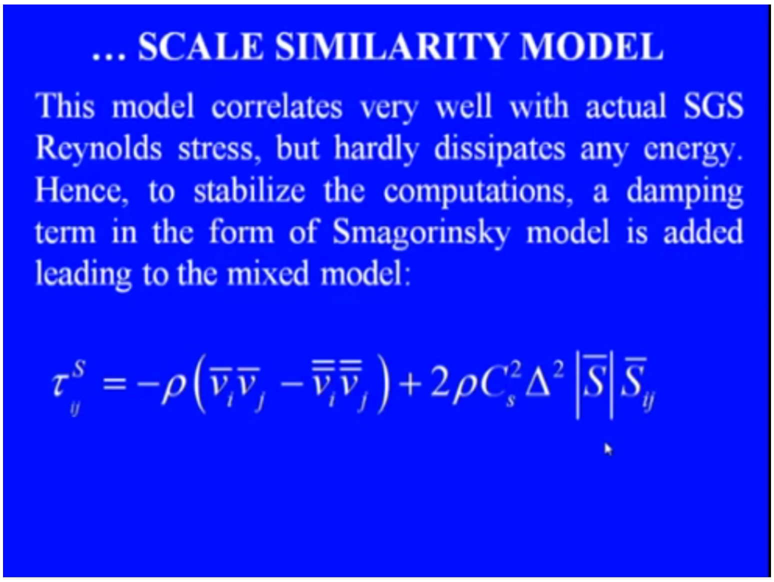


Next let us have a very brief look at scale similarity model. The basic region which Bardina et al argued were that that important interactions between resolved and unresolved scales. They would involve what? They would involve the smallest resolved eddies and the largest eddies of unresolved scales. So their interaction is the one which is most important and maybe the isotropic model may not be able to incorporate that interaction.

So what we should do is there is a similarity, which exists between these smallest resolved scale and still smaller unresolved scales. So if we can use a double filtering that can help us out so that is based on this argument. Bardina et al gave which is scale similarity model. There is tau ij S=-rho vi bar vj bar-vi double overbar*vj double overbar where double overbar indicates the quantity that has been filtered twice that is we have applied the filtering twice to determine the values of these velocity components.

Now there is some problems with this model that though it correlates very well with actual SGS Reynolds stress, but it hardly dissipates any energy. So turbulent flow simulations usually blow up if you use this model.

(Refer Slide Time: 45:42)



Hence to stabilize the computations additional damping is introduced in the form Smogorinsky model. So we add that additional damping and that leads us to what we call a mixed model. The first part is our scale similarity model and the second one is similar to our Smogorinsky model. So this mixed model is tau ij S=-rho vi overbar vj overbar-vi double overbar vj double overbar+2 rho CS square delta square modulus of S bar*S bar ij.

And this scale similarity model with enhanced damping it works pretty well and it gives us what we call stable large eddy simulation model. There are many more models, which are available, but we do not have time to discuss them in detail in this course. So I would refer you to look at some of these book references for turbulent flows for the most recent turbulence models including the more advanced large eddy simulation model.

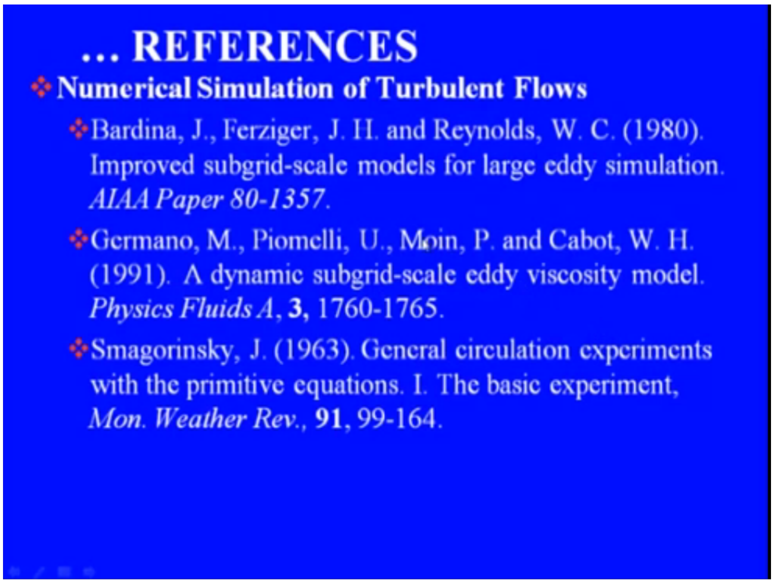
Please have a look at the book by Lesicur et al turbulence fourth edition, this book was published in 2008 by Springer.

(Refer Slide Time: 46:56)

Similarly for details of turbulent flows including different scales and turbulence models you can have a look at turbulent flows book by Pope et al. For numerical simulation related thing, Chung's book also gives us quite some details similarly you can have a look at the book by Ferziger et al on computational method for fluid dynamics and this Versteeg and Malalasekara's book on finite volume method that is introduction to computational fluid dynamics.

The finite volume method this has got one full chapter on numerical simulation of turbulent flow that is on turbulence modeling it also gives you in bit more detail many RANS models and some earlier models and few indications of which model to use in what situation. What are the advantages and disadvantages of different models and which is supposed to be used in certain flow situations?

(Refer Slide Time: 48:10)



So please have a look at that chapter in this book. Next references on what we sort today related to large eddy simulation, we referred this scale similarity model of Bardina et al, this Bardina, Ferziger and Reynolds which was derived in 1980 called improved subgrid scale model for large eddy simulation. Similarly, we briefly mentioned dynamic model so this dynamic model it computes this CS coefficient dynamically depending on the local situation.

So it is called dynamic subgrid scale eddy viscosity model and Smogorinsky model which was original or what we call initial large eddy simulation model. As far as this particular course is concerned, we would stop here. We are not going to discuss more turbulent models in details. So further details as I mentioned please have a look at these 2 books, the book by Lesicur and book by Versteeg and Malalasekara, which will give you a lot more information regarding the turbulent flow simulations.

So for this module we stop here. In the next module, we can see few practical aspects of CFD linked with grid generation and what else we need to do to validate our CFD simulations.