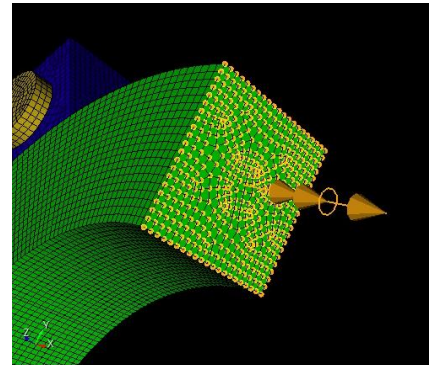


Using Coreform Cubit in a CFD workflow

Presented by
Yoshiteru Tokuyama
www.jpmandt.com

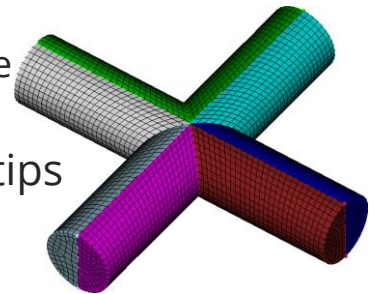
Introducing Yoshiteru Tokuyama

- CFD engineer since 1998
- Founded M&T (www.jpmandt.com) to assist Fluent users to transition from Gambit to Cubit
- Platinum Partner with Coreform (previously csimsoft) since 2008
- Based in Tokyo with many clients across industries
- Leading expert in using Cubit for CFD applications
- Recently granted a patent in Japan for efficient CFD workflows



Today's webinar

- Part 1 – Primer in Computational Fluid Dynamics
 - Which meshes work best
 - How to consider physics during model preparation
 - The importance of aligning elements with the streamline
- Part 2 – Using Coreform Cubit for CFD: 4 powerful tips
 - Visualizing the mesh before you begin
 - The 5 go-to Cubit operations and 4 entity types
 - Journaling to overcome changing requirements
 - The power of “**all**” in Cubit



Webinar objectives

- Increased product quality
 - Design more reliable products and increase corporate value by understanding CFD accuracy
- Reduced training costs
 - Techniques presented today were selected from over a decade of training on Cubit for CFD
 - Check out many more Cubit videos for CFD at coreform.com and jpmandt.com
 - Additional CFD and FEA training is available through Coreform and M&T
- Reduced labor costs
 - Better understanding Coreform Cubit can lead to dramatically reduced meshing costs: up to 100x more efficient than your current system.

Other webinar guidelines

- Questions may be asked in the webinar's Q&A section
 - We will answer as many as possible during the presentation
 - Others will be saved until the end
- Tokuyama's presentation and notes will be made available
 - Email sent to all registered webinar attendees
 - Posted at forum.coreform.com
- Additional follow and questions should be directed to forum.coreform.com
 - Tokuyama will address on the forum

Conclusion

Today we learned the following:

- Part 1 – Primer in Computational Fluid Dynamics
 - Which meshes work best
 - How to consider physics during model preparation
 - The importance of aligning elements with the streamline
- Part 2 – Using Coreform Cubit for CFD: 4 powerful tips
 - Visualize the mesh before you begin
 - The 5 go-to Cubit operations and 4 entity types
 - Journaling to overcome changing requirements
 - The power of **“all”** in Cubit

Now it is time for your questions!

Guidelines for Creating Mesh for CFD and Importance of Quickly Mastering CUBIT

**- Once you understand these concepts,
you will be able to quickly create the mesh you want. -**



sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

Guidelines for Creating Mesh for CFD and Importance of Quickly Mastering CUBIT

- Once you understand these concepts, you will be able to quickly create the meshes you want. -

What is the mesh scheme with the best accuracy for CFD?

The collage features several key elements:

- Search Results:** Screenshots of Google searches for "All Hex Mesh CFD", "Tetra Mesh CFD", and "Polyhedral mesh".
- Technical Diagrams:** Diagrams from ANSYS Meshing and ICEM CFD showing meshing techniques for various geometries like a teapot and a helicopter.
- Mesh Visualizations:** 3D renderings of different mesh schemes: a structured hexahedral mesh (SUBMAP), a structured mesh with curved boundaries (MAP), a tetrahedral mesh (TETRA), and an irregular polyhedral mesh (Polyhedral).
- Text Labels:** Large text labels "SUBMAP", "MAP", "TETRA", and "Polyhedral" are placed over their respective mesh visualizations.



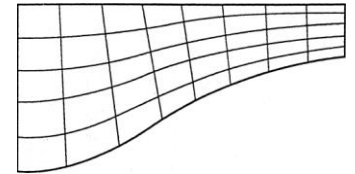
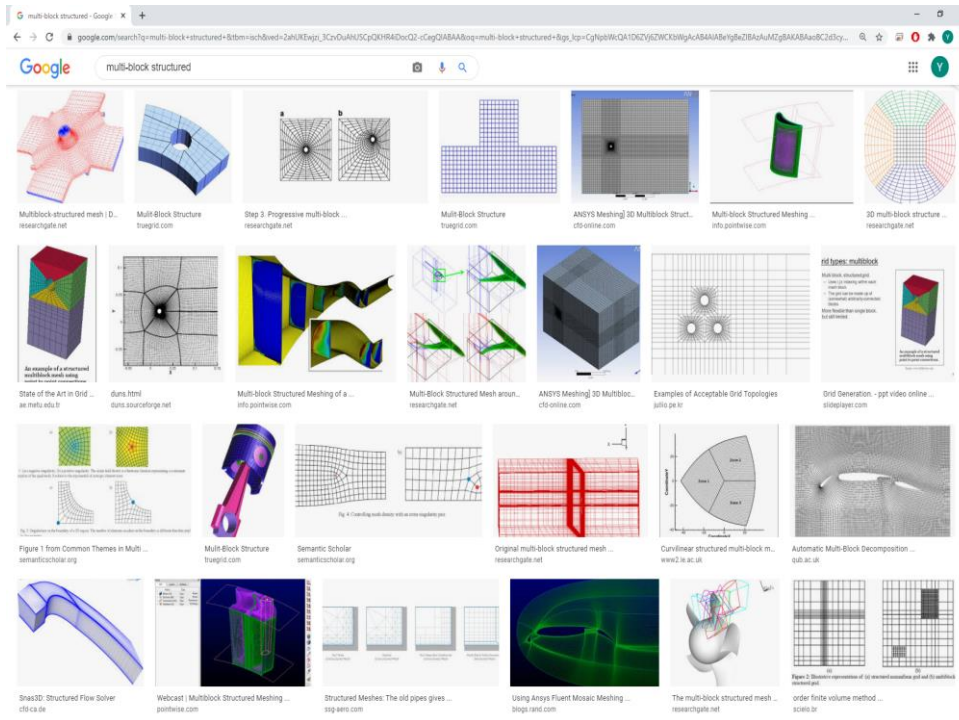
Hi! My name is Yoshiteru Tokuyama.

What do you think is the mesh scheme with best accuracy for CFD?

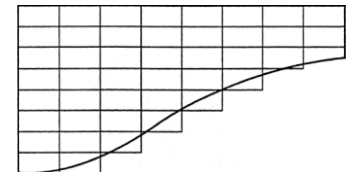
Is it Submap, Map, or Tetra? Or Polyhedral?

tokuyama@jpmant.com

Doesn't CUBIT automatically create all hexa-mesh?



Body-Fitted Coordinates



Cartesian Coordinates

M&T Multi-blocked Structured Mesh

sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

Customers often ask, "Doesn't CUBIT automatically create all hex mesh?"

Also, "Can't you create a beautiful hex mesh of the entire model domain?" The question is also asked.

The image of a beautiful mesh in the mind of a customer who asks such a question is more likely to look like a figure (multi-block structured), isn't it?

The image is that the surface of the object is beautifully meshed with BFC (Boundary-Fitted Coordinates), and the mesh topology of the structured grid is attached near the object.

Is possible to create multi-blocked structured mesh for any geometry?

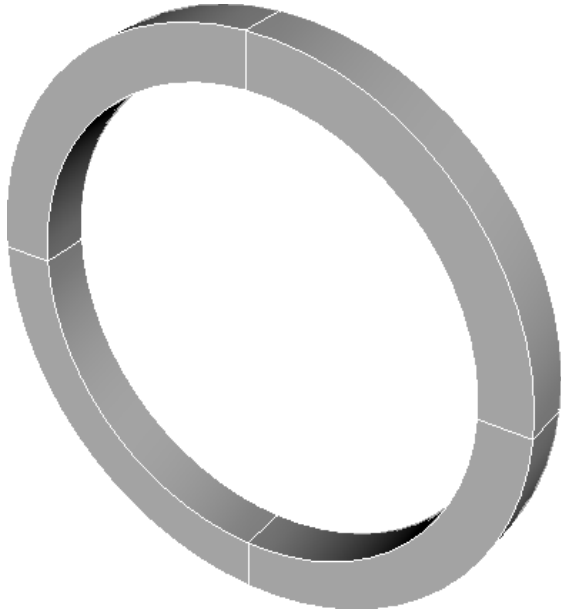


Fig. (a)

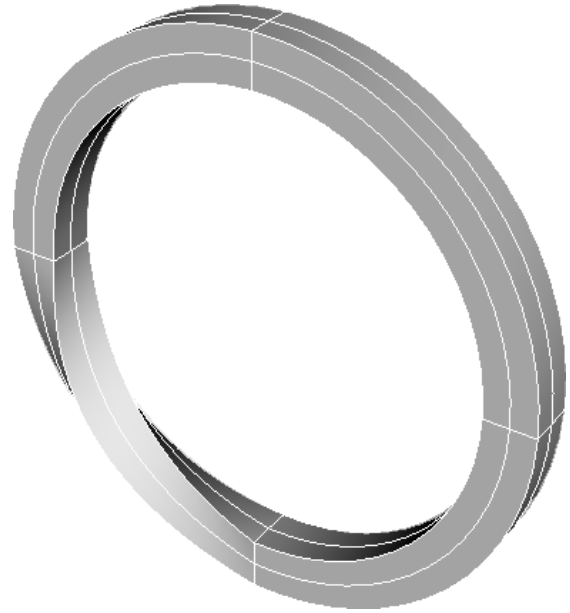


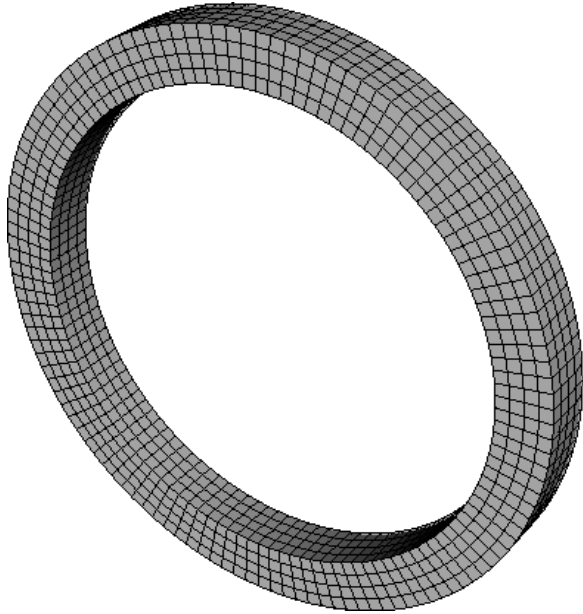
Fig. (b)



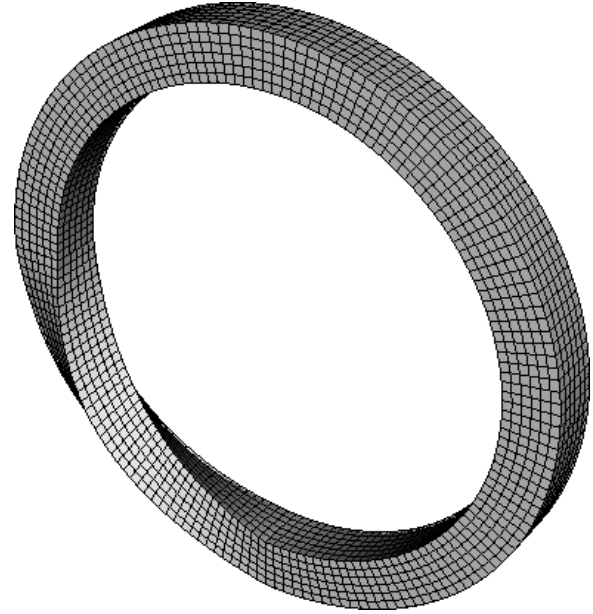
sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

Is it possible to create such a mesh for the geometry in figure (a) and figure (b)?

Is possible to create multi-blocked structured mesh for any geometry?



Solid domain
Map/Sweep type meshable



Solid domain
Map/Sweep type meshable



sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

Yes, you can. Is it easy?

Is possible to create multi-blocked structured mesh for Space outside the solid?

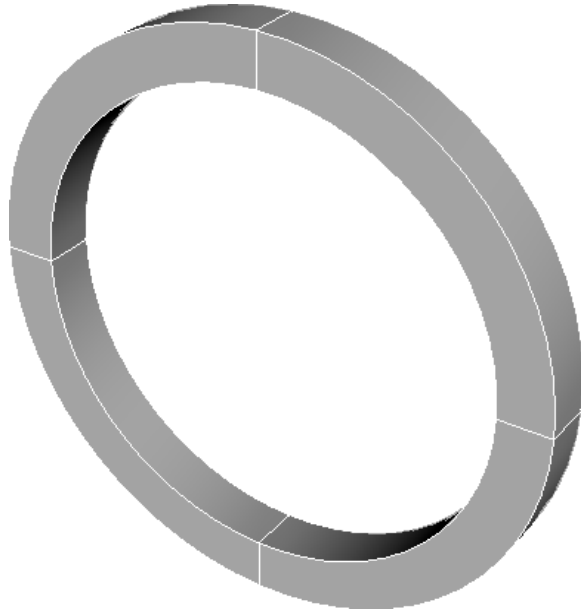


Fig. (a)

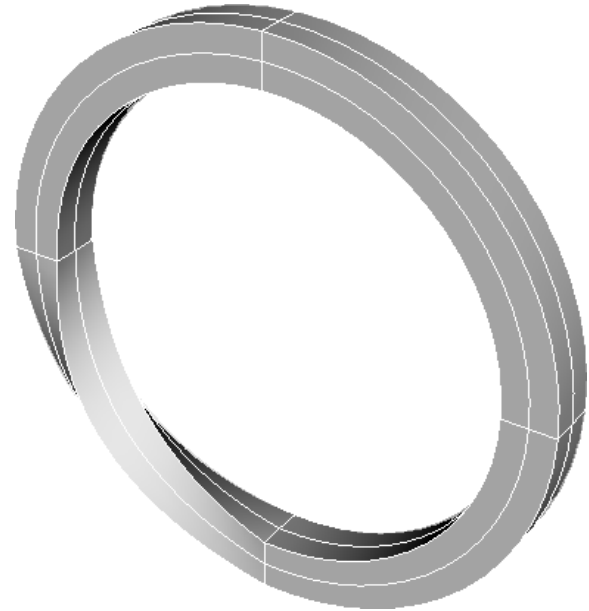
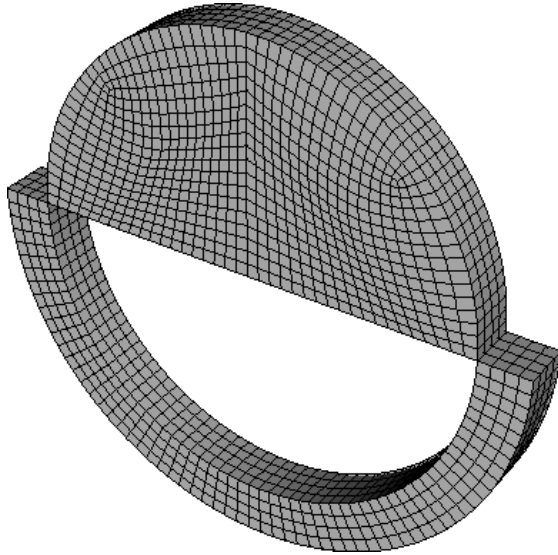


Fig. (b)

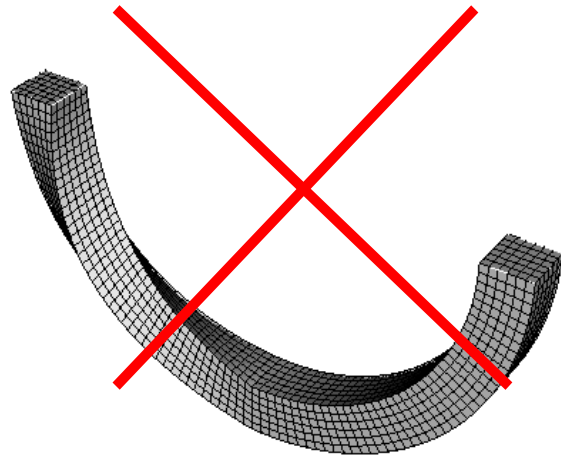


So, can we create such a mesh for the space surrounding this solid shape?

Is possible to create multi-blocked structured mesh for Space outside the solid?



Air domain
Map/sweep type Meshable



Air domain
Map/sweep type mesh impossible



The answer is;

(a) Yes.

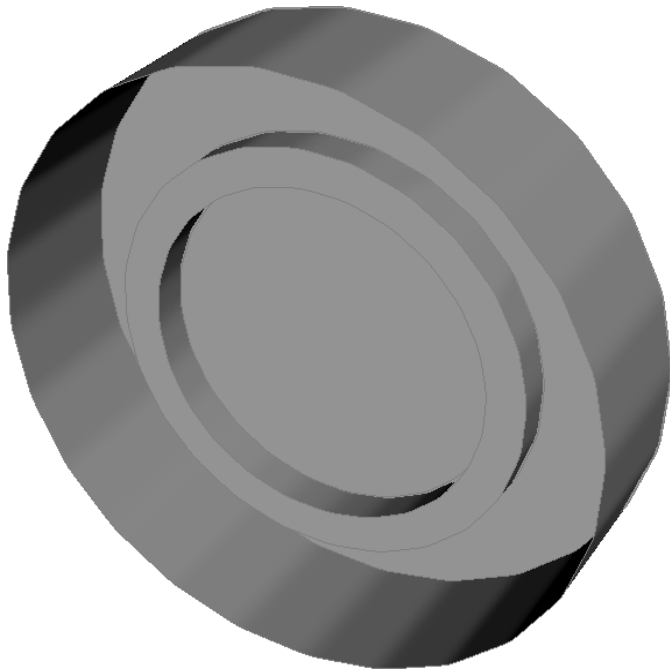
But (b) is not possible.

You know why, right?

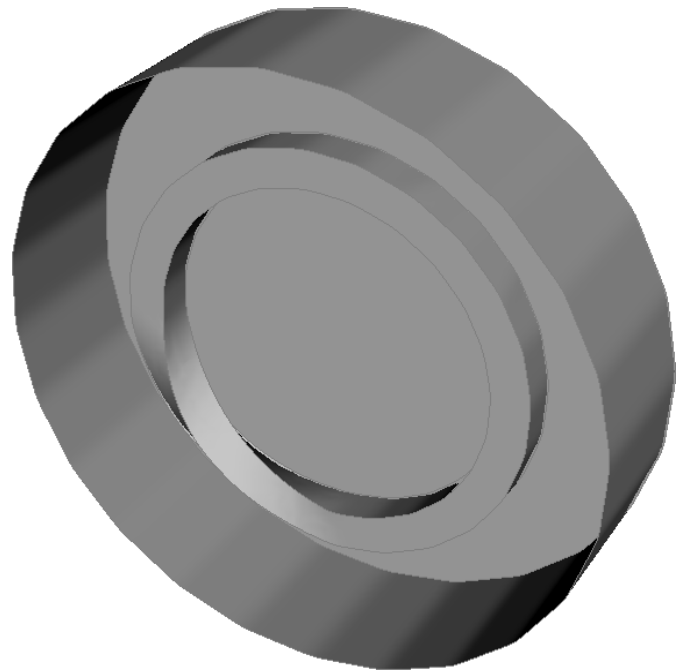
The front and back of this structure are turned upside down.

Therefore, it is not possible to create a continuous mesh in a BFC-type multi-block structured grid.

Is possible to create all hex mesh for Space outside the solid?



Air area
Is Sculpt type mesh possible ?



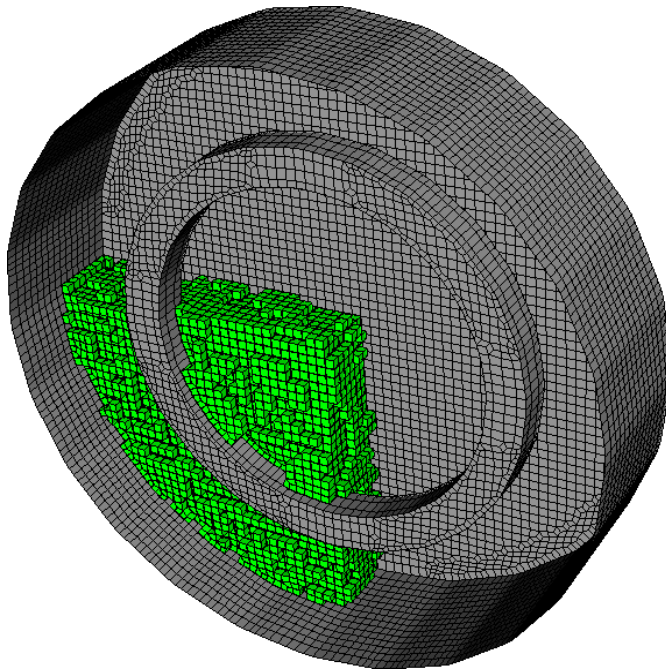
Air area
Is Sculpt type mesh possible ?



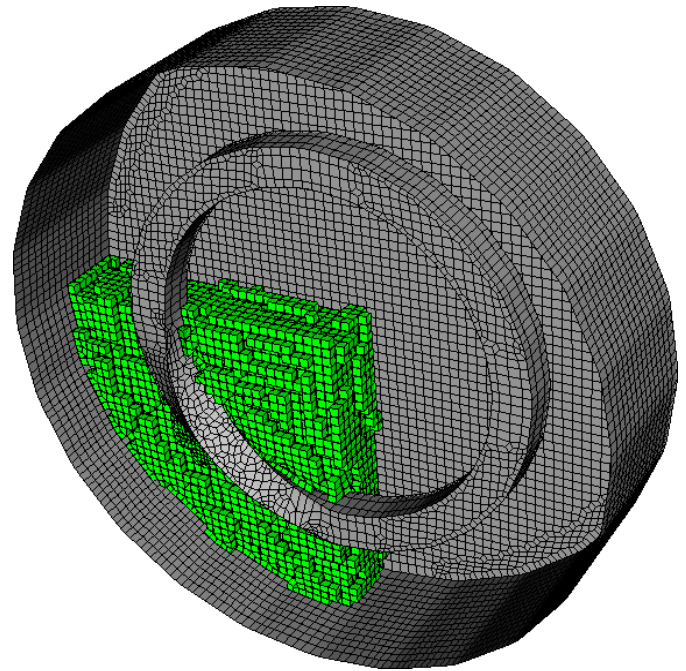
sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

So, can we create an “All Hex Mesh” for the space surrounding this shape?

Is possible to create all hex mesh for Space outside the solid?



Air area
Sculpt type mesh possible



Air area
Sculpt type mesh **possible**



draw hex with $x_coord < 0$ and $y_coord < 0$ and $z_coord < 0$ add

sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

The answer is yes.

BFC-type multi-block structured grid just can not mesh continuously,
but using algorithms like Sculpt,

ALL Hex Mesh can be easily created.

However, the number of mesh elements required to represent a shape is very large.

Commonly used features in Fluent and the relationship between element count and memory.

Table A : Frequently used features in FLUENT

Solver	Steady	unsteady		
Viscous Model	Laminar	k-e (RANS)	LES	
Spatial Discretization	1 st Order Upwind	2 nd Order Upwind	QUICK	Central Differencing

Table B : Memory capacity required for FLUENT calculations

Memory	Number of elements	Resolution per direction	Corresponding screen resolution
2G	1 x 100 x 100 x 100	$1^{(1/3)} \times 100 = 100$	sQCIF(Sub-QCIF)
8G	4 x 100 x 100 x 100	$4^{(1/3)} \times 100 = 159$	
16G	8 x 100 x 100 x 100	$8^{(1/3)} \times 100 = 200$	
32G	16 x 100 x 100 x 100	$16^{(1/3)} \times 100 = 252$	QVGA (Quarter-VGA)
64G	32 x 100 x 100 x 100	$32^{(1/3)} \times 100 = 317$	
128G	64 x 100 x 100 x 100	$64^{(1/3)} \times 100 = 400$	
256G	128 x 100 x 100 x 100	$128^{(1/3)} \times 100 = 504$	VGA
512G	256 x 100 x 100 x 100	$256^{(1/3)} \times 100 = 635$	SVGA
1T	512 x 100 x 100 x 100	$512^{(1/3)} \times 100 = 800$	XGA
2T	1024 x 100 x 100 x 100	$1024^{(1/3)} \times 100 = 1008$	XGA+



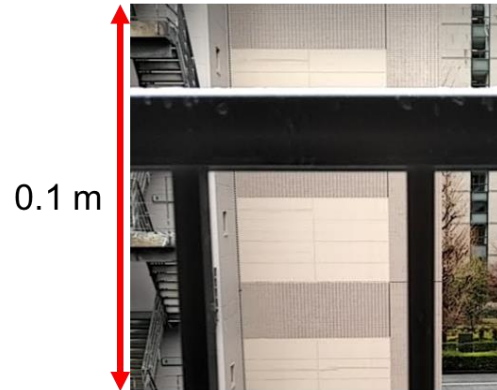
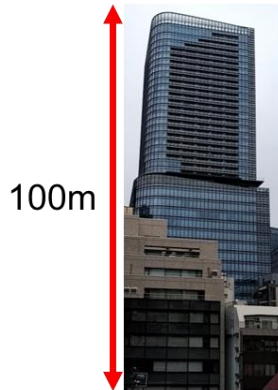
The typical features available in FLUENT and the relationship between computer and memory are summarized in Tables A and B.

As you can see from this table, a user who needs to perform CFD simulations on a computer with 16G of memory can only express the data with a resolution of 200 per direction. That is only 200.

A resolution of 1000 mesh elements means that each mesh is only 1mm in a 1m analysis domain, which is nearly impossible for your PC to represent a 1mm object in a 1m analysis domain.

A resolution of 100 mesh elements means that each mesh is only 1cm in a 1m analysis domain, and the average PC today can only represent objects that are 1cm in a 1m analysis domain.

Simulation objects of widely varying scales



$$100 / 0.1 = 1000$$

$$1000 \times 1000 \times 1000 = 1,000,000,000$$

It takes 2T of memory just to resolve the minimum size.



sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

A customer in the construction industry once asked me to simulate the effect of a balcony railing (1cm to 10cm) on window glass in a high-rise apartment building (let's say 100m).

You can already see that it is impossible to do this analysis by direct simulation in the current computing environment, right?

(If this were represented by an evenly spaced orthogonal mesh, a huge number of elements would be required. No amount of memory will suffice.)

So, it is important to design a mesh that can reproduce the phenomenon more accurately with a realistic mesh size.

Imagine a mesh that you think would work best for this shape!

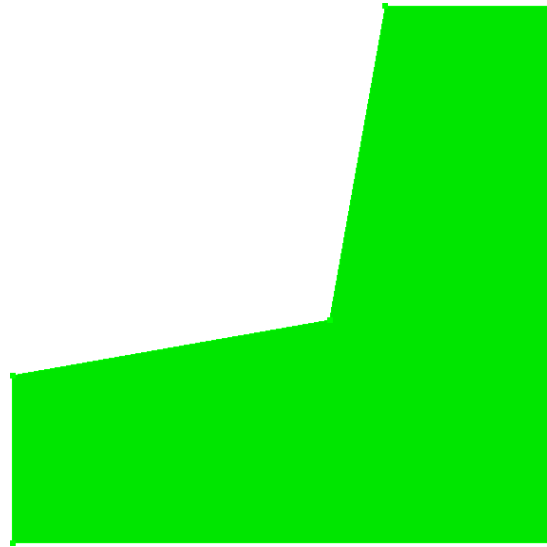


Fig. (a)



Today, I will talk about fluid and heat transfer simulation as an example.

Also CFD is assumed to be an unstructured CFD solver for finite volume methods, such as FLUENT. For structured grid solvers, the story is a little different.

I would like to start with a simple question.

What kind of mesh would you create for this kind of geometry?

Think about it.

I do not know what kind of mesh you have in your mind, but one thing I can tell you is that the meshes you imagine will never match exactly.

Is there a mesh topology in this list that you have considered?

Which of the following mesh has a small number of elements and good mesh quality?

Do you think this mesh is the best for CFD simulation? Think about it for a moment.

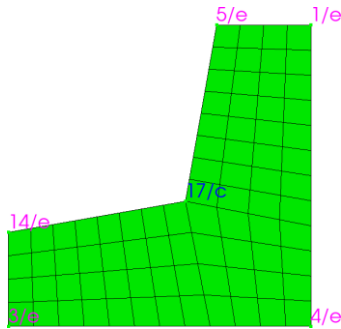


Fig. (a) SUBMAP 80 elements Shape 0.9734

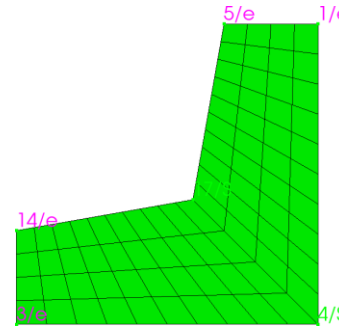


Fig. (b) MAP 108 elements Shape 0.8224

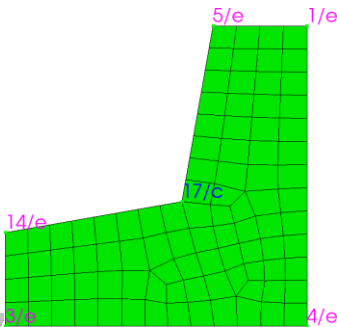


Fig. (c) PAVE 93 elements Shape 0.9666

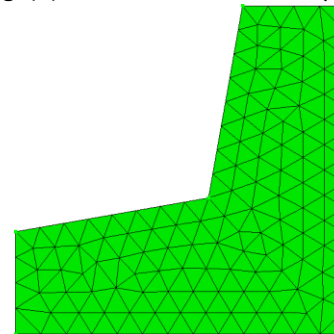


Fig. (d) TRI 238 elements Shape 0.9786

les@coreform.com
ma@jpmandt.com

For example, some of you may have imagined a mesh topology close to the structured grid as shown in figure (a), some of you may have imagined a mesh topology of the BFC type structured grid as shown in figure (b), and some of you may have imagined a mesh as shown in figure (c) and (d).

Which of these mesh has a small number of elements and good mesh quality?

It is the type of mesh in figure (a).

Do you think this mesh is the best for CFD simulation? Think about it for a moment.

The answer is, "Sometimes it's suitable, and sometimes it's not".

If the mesh quality is good, the simulation will be stable.

However, good mesh quality does not guarantee the best simulation accuracy or the best results at all.

Do you think the best CFD mesh can be created before the simulation is done?

- The answer, in general, is “No”.
 - If you have the best CFD mesh before you simulate, then you don't need to do the simulation.
 - The best CFD mesh is the simulation result itself.
 - A good mesh is one that can reproduce the real phenomena with high resolution.



sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

Here is another question. Do you think the best CFD meshes can be created before the simulation is done?
The answer, in general, is “No”. If you have the best CFD mesh before you simulate, then you do not need to do the simulation. The best CFD mesh is the simulation result itself. A good mesh is one that can reproduce the real phenomena with high resolution.

Skills required to create a good mesh

- The user must be able to imagine the desired hexa-mesh in order to reproduce real phenomena with high accuracy.
- The user must be familiar with physics, i.e., the user must have an image of the flow before creating the mesh.
- If the user does not have an image of the physical phenomenon, the mesh cannot be created to reproduce the phenomenon with high accuracy.



sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

The skills and points needed to create a good mesh are the three listed here.

1. The user must be able to imagine the desired hex mesh in order to reproduce real phenomena with high accuracy.
2. The user must be familiar with physics, i.e., the user must have an image of the flow before creating the mesh.
3. If the user does not have an image of the physical phenomenon, the mesh cannot be created to reproduce the phenomenon with high accuracy.

Advection-diffusion equation

$$\frac{\partial}{\partial t} \int_V \rho \phi dV + \int_S \rho \phi \mathbf{v} \cdot d\mathbf{S} = \int_S \Gamma_\phi \nabla \phi \cdot d\mathbf{S}$$

The most important thing to obtain better calculation results in CFD is

- Accurate reproduction of advection phenomena
- Accurate reproduction of diffusion phenomena



sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

In order to create the best CFD mesh, there is something that the user needs to understand.

CFD is computational fluid dynamics, where the Navier-Stokes equations are solved numerically.

A more generalized version of the Navier-Stokes equations is called the advection-diffusion equation.

The advection-diffusion equation for momentum, or velocity, is the Navier-Stokes equation.

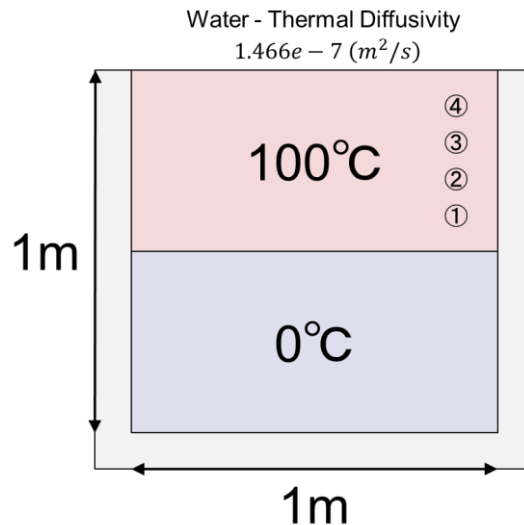
I think the most important thing to obtain better calculation results in CFD is to reproduce the velocity field more accurately, whether it is a steady-state analysis or an unsteady analysis. In other words, it is important to accurately reproduce the advection phenomenon. The next important thing is to reproduce the diffusion correctly.

1. Accurately reproduction of advection phenomena
2. Accurately reproduction of diffusion phenomena

Advection-diffusion

Here's a question: after 1 minute, what is the temperature of the place ①?

A: 100 °C, B: 50 °C, C: about 90 °C



sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

In a flow field, physical quantities are transported by advection and diffusion. For example, it is easier to understand heat transport if we consider it as an unsteady phenomenon. Consider the situation as shown in the figure. The temperature of the water in the bathtub is transported by advection and diffusion.

The upper half of the water in this bath is 100 degrees Celsius, and the lower half is 0 degrees Celsius. Think of a situation where the flow of water is completely stopped.

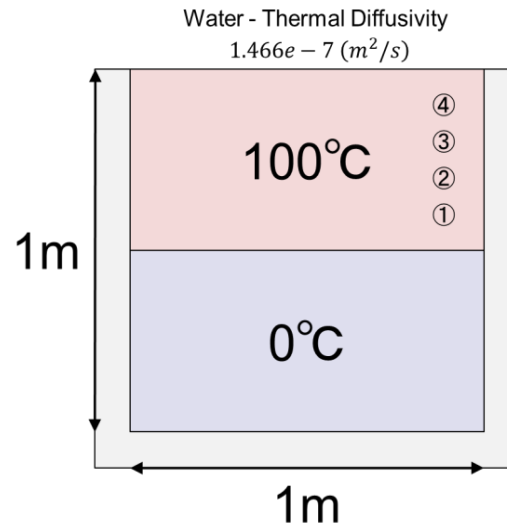
Here is a question: after 1 minute, what is the temperature of the place (1)?

A: 100 degrees Celsius, B: 50 degrees Celsius, C: about 90 degrees Celsius

The answer is A.

Advection-diffusion

Estimation of temperature spread by temperature diffusion		
	t [s]	$l_\sigma = \sqrt{Dt}$ [m]
1s	1	0.0003829
1min	60	0.0029658
1 hour	3600	0.022973
1 day	86400	0.1125444



sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

Almost, the temperature will not mix. This can be easily estimated, as shown in the table.

It will take a few days for the temperatures to fully mix.

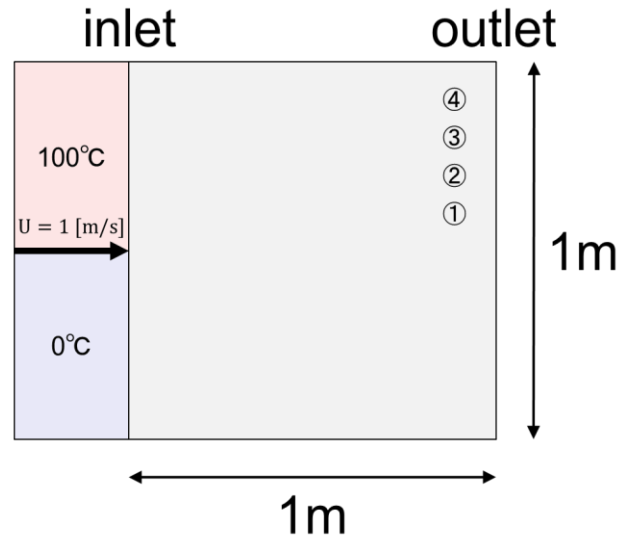
There are many people in the world who are considered to be CFD experts, but the only real CFD experts are actually those who understand the answer to this question easily enough to understand diffusion specifically.

Conversely, if you do not know the answer to this question, you do not even understand diffusion. If you do not understand diffusion, it is difficult to get the right answer in CFD.

Advection-diffusion

What is the temperature of ① near the outlet?

A: 100 °C , B: 50 °C, C: about 90 °C



sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

I would like to ask a question in a different way.

Consider the situation as shown in the figure. Water of 100 degrees Celsius and 0 degrees Celsius is flowing at 1m/s from the inlet. What is the temperature of the place (1) near the outlet?

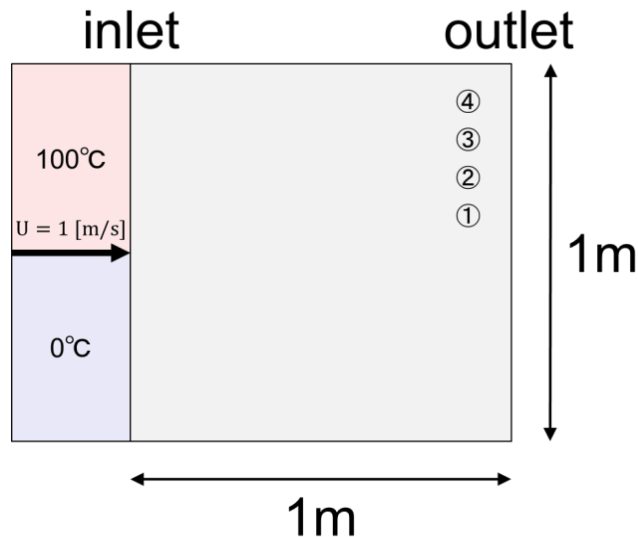
A: 100 degrees Celsius, B: 50 degrees Celsius, C: about 90 degrees Celsius

The answer is A. Almost, the temperature does not mix. The water at the inlet will be at the outlet in one second. In other words, it is the same as the previous question, "How much will it mix after one second? It hardly mixes at all.

Advection-diffusion

What is the temperature of ① near the outlet?

A: 100 °C , B: 50 °C, C: about 90 °C



sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

As you can see from this simple question, diffusion alone does little to mingle physical quantities. The most effective way to move physical quantities is through advection.

You may empirically feel that the water and hot water are mixing more quickly. That the temperature in the bathtub is mixing more quickly and is 50 degrees Celsius. That is because the stirring (by advection) creates a lot of eddies of various sizes (local point of view, advection), and the effect of the eddies promote heat diffusion. This is called turbulent diffusion. From an unsteady, instantaneous, and local point of view, it can be thought of as advection and diffusion. If you look at it in a steady state, you can think of it as a turbulent diffusion phenomenon.

Temperature carrying phenomenon

Scale	Large	Medium	Small
Unsteady Instantaneous - Local	advection	-	Diffusion
Steady	advection	Turbulent diffusion	Diffusion

If a person moving relative to the flow sees this phenomenon, the effect of advection becomes zero. In other words, the only phenomena that carry temperature will be turbulent diffusion and diffusion phenomena. If there is no turbulence in the flow, there will be only diffusion.



Scale	Medium	Small
Unsteady Instantaneous - Local	-	Diffusion
Steady	Turbulent diffusion	Diffusion



Let me try to put this in perspective.

I said that physical quantities in a flow field are transported by advection and diffusion, and I will add the phenomenon of turbulent diffusion to this. This is summarized in the table.

If a person moving relative to the flow sees this phenomenon, the effect of advection becomes zero. In other words, the only phenomena that carry temperature will be turbulent diffusion and diffusion phenomena. If there is no turbulence in the flow, there will be only diffusion.

Temperature carrying phenomenon in CFD

		Influence	Grid	Turbulence model	Thermal diffusion
		Scale	?	Small ~ Medium	Small
Unsteady	Laminar	les	False diffusion	-	Diffusion
	Turbulence			Turbulent diffusion (small)	
Steady	Laminar	laminar		-	
	Turbulence	k-e (RANS)		Turbulent diffusion	



sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

In fact, there is another phenomenon in CFD that can carry temperature. It is called false diffusion. Depending on the discretization scheme, this false diffusion can instantly carry the temperature from the element width to about three times the maximum element width. This effect is almost the same for both hex and tetra mesh with the same discretization scheme for the same width of false diffusion. False diffusion occurs in both hex mesh and tetra mesh. A hex mesh does not mean that there is no false diffusion and the accuracy is good. False diffusion disappears only when the velocity field and the mesh are parallel. In other words, there is no false diffusion only in the mesh along the streamline. In other words, the best mesh is the one that follows the streamlines! In terms of computational cost, the cost of creating a mesh along the streamlines is much smaller than the cost of reducing the mesh size.

Temperature carrying phenomenon in CFD

		Influence	Grid	Turbulence model	Thermal diffusion
		Scale	?	Small ~ Medium	Small
Unsteady	Laminar	les	False diffusion	-	Diffusion
	Turbulence			Turbulent diffusion (small)	
Steady	Laminar	laminar		-	
	Turbulence	k-e (RANS)		Turbulent diffusion	



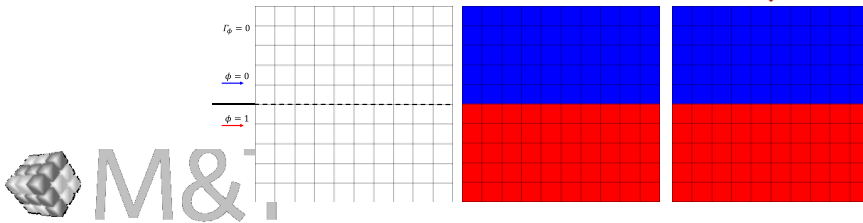
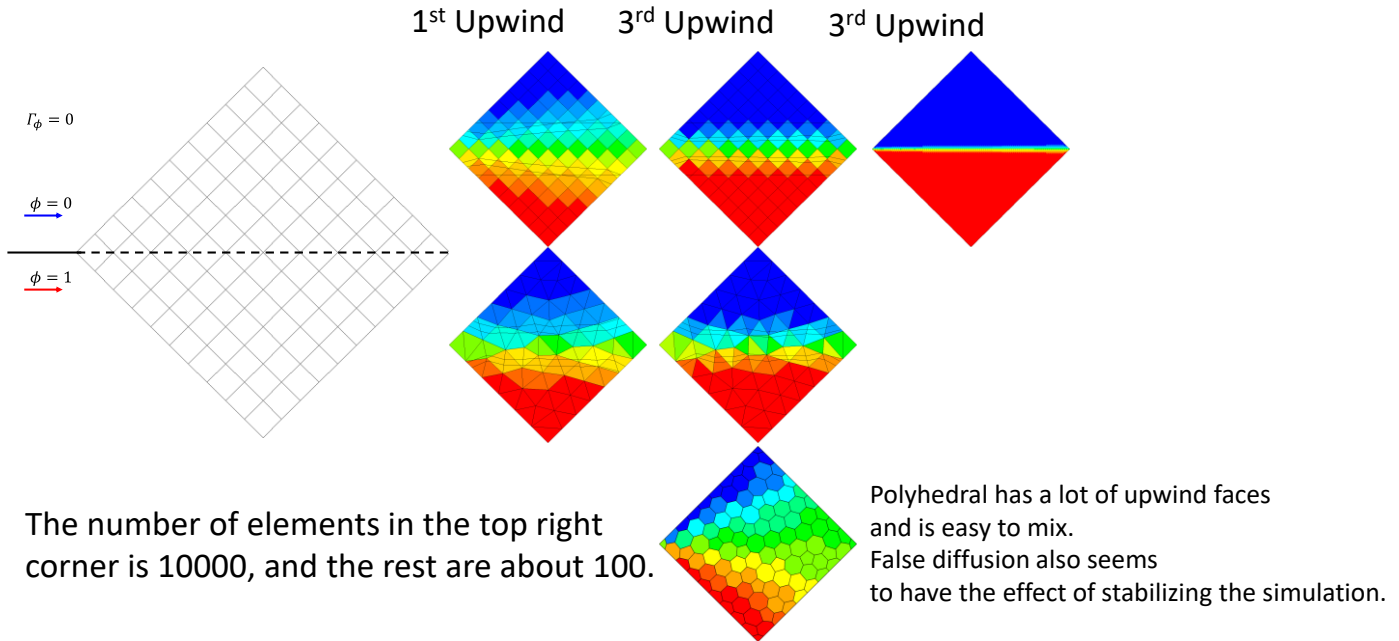
sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

The effect of this false diffusion on the accuracy of the simulation can be neglected in many cases, except near the wall, where the effect of turbulent diffusion by the turbulence model is significant.

The effect of false diffusion on the accuracy of the analysis is in the region of laminar flow, where heat is transferred only by the effect of the thermal diffusivity. The problem of false diffusion is also a major issue when solving species transport, which involves combustion. Gases that should not mix, but due to false diffusion, they do! As a result, they burn well!

Also, the temperature inside an enclosure that generates heat, and contains a lot of electrical components, cools well!

False diffusion Example



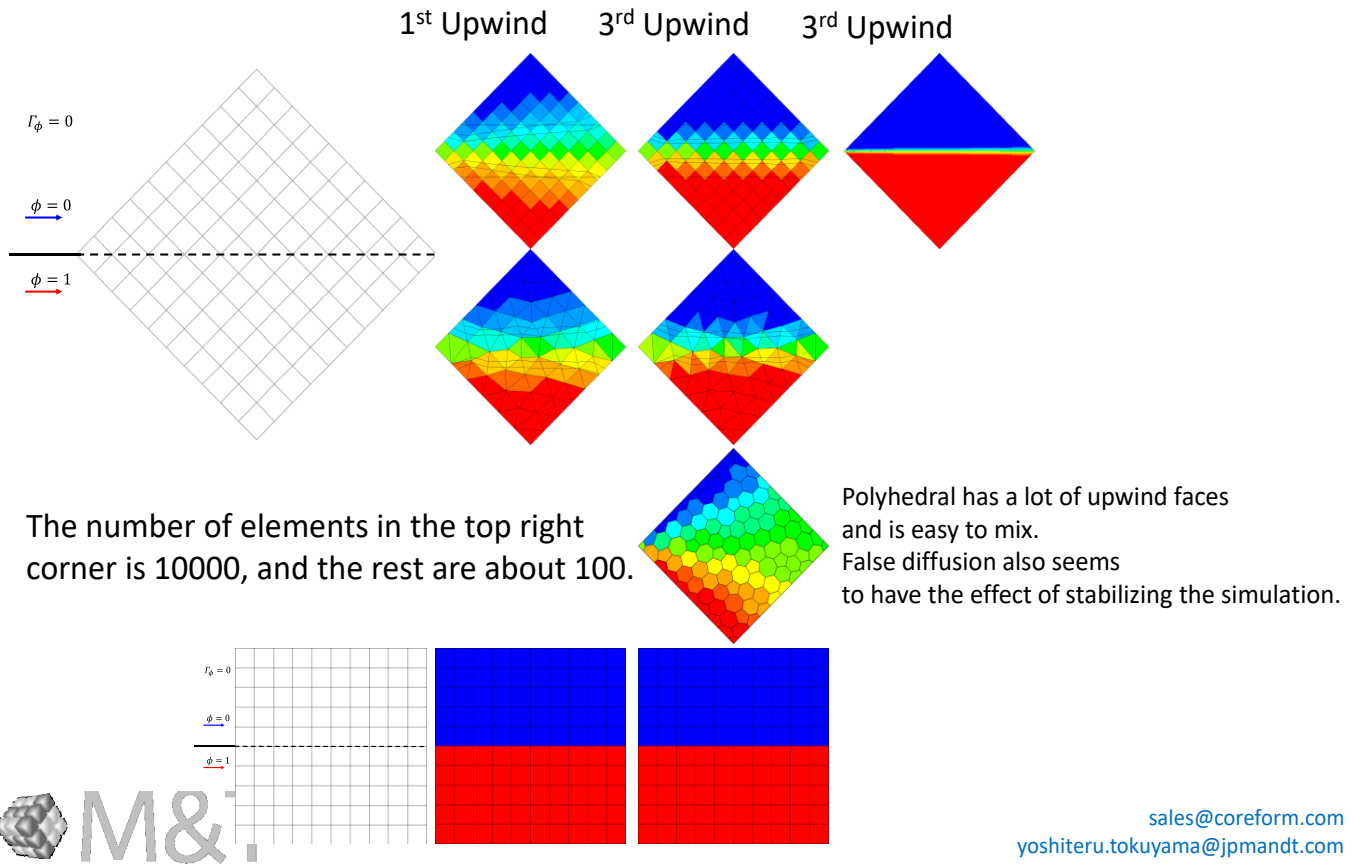
sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

Now let's look at a concrete example.

Here, a gas with a diffusion coefficient of zero is flowing at 1 m/s to the right. The temperature in the upper half is 0 degrees Celsius and the temperature in the lower half is 1 degrees Celsius. The answer to this question is very simple. Since the diffusion coefficient is 0, the answer will be all 0 in the upper half and all 1 in the lower half, with no mixing at all.

The elements number is adjusted to be roughly 100. Only the top right geometry has 10,000 elements.

False diffusion Example

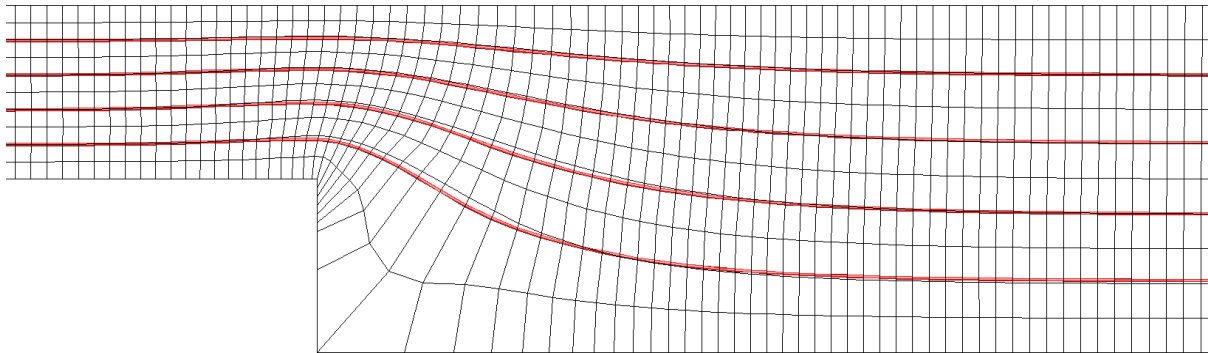


When the direction of the flow and the direction of the mesh do not coincide for both quad and tri meshes, we can see that false diffusion occurs in both discretization schemes. It can be seen that the false diffusion does not disappear even when the mesh width is reduced to 1/10, i.e., when the number of meshes is increased by a factor of 100. On the other hand, in the case where the flow direction and mesh direction are aligned, there is no false diffusion at all, regardless of the discretization scheme.

These results are due to the nature of CFD's unique upwind differencing scheme, and should be taken into account when designing a CFD mesh!

Guide line for CFD Mesh and Example

Method 1: The mesh should be created along the velocity field. That is, it should be created along a streamline.



Method 1: The mesh should be created along the velocity field.

In other words, it should be created along a streamline.

In CUBIT, all you have to do is sweep the mesh in the streamline direction!

I can hear you say, "I don't know where the streamlines are!"

Guide line for CFD Mesh and Example

From this figure (a), we can clearly see that the mesh created along the streamlines has suppressed false diffusion.

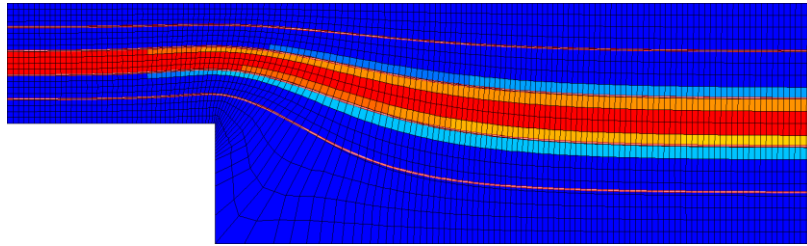


Fig. (a) 1st Upwind

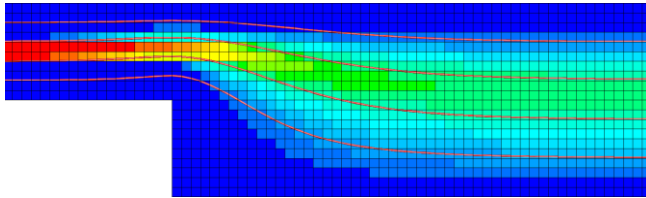


Fig. (b) 1st Upwind

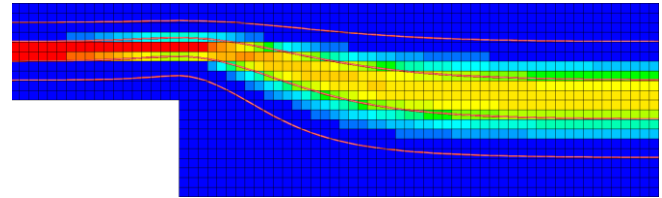


Fig. (c) 3rd Upwind



The effect of false diffusion on the mesh created according to method 1 and on the mesh created with a normal structural grid is compared here.

The red lines shown in this figure are streamlines.

Figure (a) is created by method 1. Figure (b) is the structural grid and the discretization scheme is 1stUpwind, and figure (c) is 3rd Upwind.

From this figure, we can clearly see that the mesh created along the streamlines has suppressed false diffusion.

Conclusion

Guide line for Best CFD Mesh

1. Mesh should be created from streamlines and their vertical lines.
2. Determine the resolution of the mesh so that it can represent the curvature of the streamlines. In other words, create coarse mesh where the radius of curvature of the streamlines is large and fine mesh where the radius of curvature is small.
3. Create finer mesh for locations where the physical quantity changes abruptly, and gradually increase the size of the mesh, smoothly connecting the coarse mesh.
4. Create fine mesh where the change in physical quantity is large and coarse mesh where it is small. Where the physical quantity changes significantly, complex phenomena occur.



sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

We will refine mesh creation method 1 a bit more and put it together with other general methods.

These four methods are the meshing methods necessary to accurately capture advection and diffusion with a smaller number of elements. **Based on the mesh created in this way, the flow field is evaluated again, and by repeating this procedure, it is possible to obtain simulation results that eliminate false diffusion.** In CFD simulations, there are places where false diffusion will affect the accuracy of the simulation and places where it will not. The user should understand this in advance and design a mesh that does not cause false diffusion in the right place at the right time.

Conclusion

Guide line for Best CFD Mesh

1. Mesh should be created from streamlines and their vertical lines.
2. Determine the resolution of the mesh so that it can represent the curvature of the streamlines. In other words, create coarse mesh where the radius of curvature of the streamlines is large and fine mesh where the radius of curvature is small.
3. Create finer mesh for locations where the physical quantity changes abruptly, and gradually increase the size of the mesh, smoothly connecting the coarse mesh.
4. Create fine mesh where the change in physical quantity is large and coarse mesh where it is small. Where the physical quantity changes significantly, complex phenomena occur.



sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

For places where turbulent diffusion is large or where there is no change in the field, designing with a tetra mesh will not cause any particular problem. However, in places where the influence of viscosity is strong, careful mesh design is necessary.

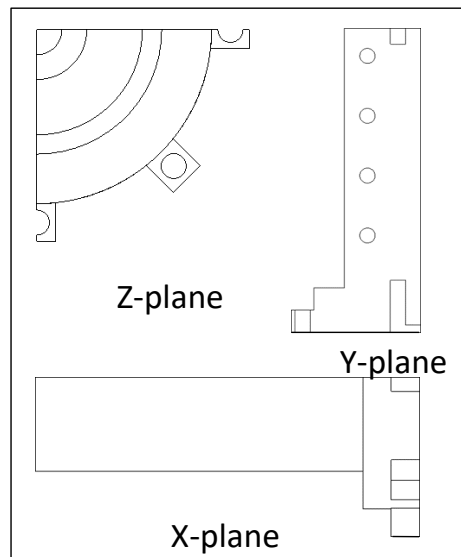
If a fully automated mesher with highly accurate simulation results is possible, it cannot be achieved by AI technology, but only by physics. This is because the best mesh depends on physical phenomena and an infinite number of boundary conditions, and there is an infinite number of fields to be obtained, with one best mesh pattern corresponding to one field.

This concludes our discussion on CFD meshing guidelines.

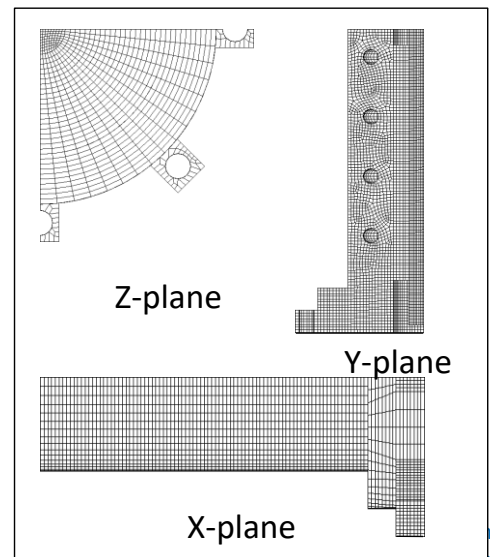
Draw, by hand, the geometry and mesh you want to create!

- Imagine what you want to do!
 - Draw the abstract image in your mind on paper in a concrete way.
 - How to draw the model
 - Blueprint of the mesh you are looking for
- Organize the steps you've taken!
- Finally, simply command CUBIT. CUBIT will follow your commands.

Geometry Image



Mesh Image



"What is important to be able to create meshes using CUBIT?"

The answer is

1. To draw the geometry you want to create by hand

And

2. To draw the mesh you want to create by hand.

Finally, simply command CUBIT to take the steps that will accomplish the blueprint. CUBIT will follow your commands.

CUBIT is more than just a tool. What is important is your blueprint!

To put it another way, as long as you have a blueprint, any tool that can draw a blueprint is good, whether you use CUBIT, ICEM, or Femap.

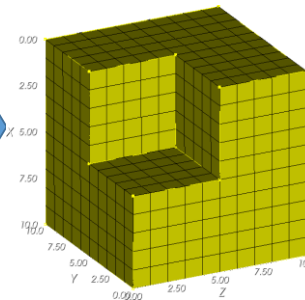
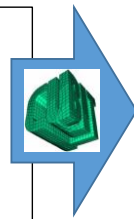
What makes CUBIT different from other tools?

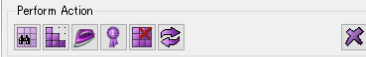
1. CUBIT understands human language.
2. If you command “mesh vol all”, CUBIT will create mesh appropriately . *With reference to default entity properties.*
3. You can easily change the size of the mesh and how it is designed(mesh scheme).

If you want to change the mesh size or scheme as you wish, just ask CUBIT to change the properties.

```
create brick x 10
create brick x 10
move volume 2 x 5 y 5 z 5
subtract volume 1 from volume 2
mesh volume all
```

CUBIT Command Language



Properties Page	
Perform Action	
	
Property	Value
General	
Id	1
Type	Volume
Name	Volume 1
Idless Signature	calc
Color	Not Set
Location	0.000000, 0.000000, 0.000000
Geometry	
Engine	ACIS
Volume	calc
Meshing	
Is Meshed	No
Number of Elements	0
Number of Nodes	0
Requested Intervals	Not Set
Requested Size	0.125
Meshed Volume	calc
Mesh Scheme	Tetmesh
Smooth Scheme	Smart Laplacian
Sizing Function	Constant

Entity Properties



What makes CUBIT different from other tools is that.

1. CUBIT understands human language.
2. If you command “mesh vol all”, CUBIT will create meshes appropriately.
3. You can easily change the size of the mesh and how it is designed.

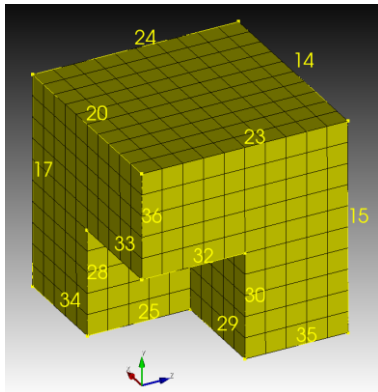
It is important to turn the vague and chaotic consciousness into an orderly, organize and clear awareness. We need to act consciously. Then you will notice that there will be a command that you can use. This is the same for everything. If you try to think like this on a regular basis, you will be able to have a better outlook on things.

To be more specific.

“1. Understand the human language.” means, i.e., if you put the procedure into words , you can create the model.

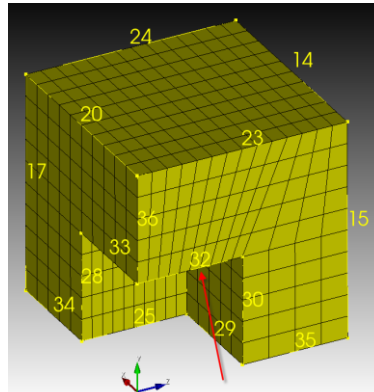
How great the journal script of CUBIT is!

Please create a mesh with a hex.



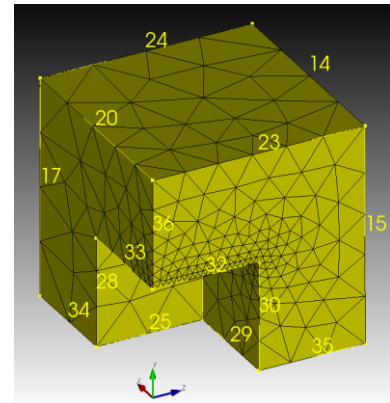
```
reset vol all
mesh volume all
```

Please make the mesh here a little smaller.



```
reset vol all
curve 32 size 0.25
mesh vol all
```

I'm sorry, but can you create it with tetra mesh?



```
reset vol all
volume all scheme tet
volume all size 2.5
curve 32 size 0.25
mesh Volume all
```



sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

Next is the mesh.

Here you can see just how amazing CUBIT's journal feature is!

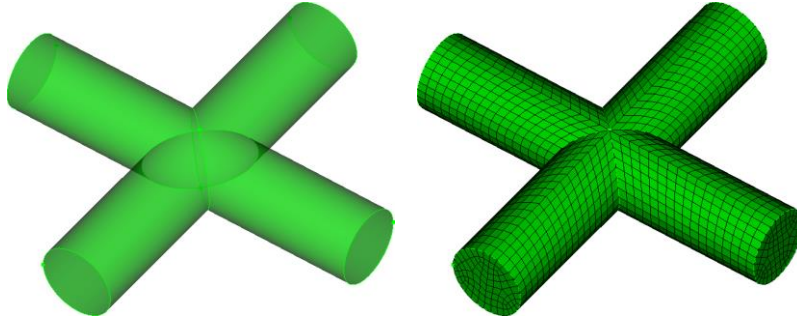
1. It can be written like a human language.
2. You can easily change the size of the mesh.

And

3. You can also easily change the size and mesh type.

Geometry type that cannot be automatically generate meshed by “Mesh Vol All”

In many cases, there are inconsistencies in the scheme automatically selected by CUBIT or the geometry type is of the Many-to-Many type.



For volumes that cannot be meshed,
The user's only job is to do one or both of the following

1. Specify a Scheme that can be meshed.
2. Decompose a Many-to-Many type volume into Many-to-One, or One-to-One, so that CUBIT can automatically select a Meshable Scheme.



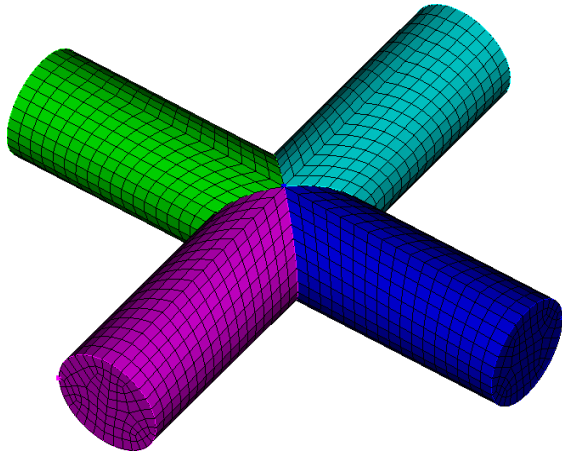
sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

As I have already mentioned, if you command “mesh vol all”, CUBIT will generate the meshes as it sees fit. It is a best effort. There are some volumes (multi-sweep type) where meshes are not generated automatically. In CUBIT, if there is a geometry that cannot be generated mesh, you just have to make the un-generated geometry cut.

In many cases, there are inconsistencies in the scheme automatically selected by CUBIT or the geometry type is of the Many-to-Many type.

Let's do some prep work for “Mesh Vol All” !

Decompose & Sweep Setting



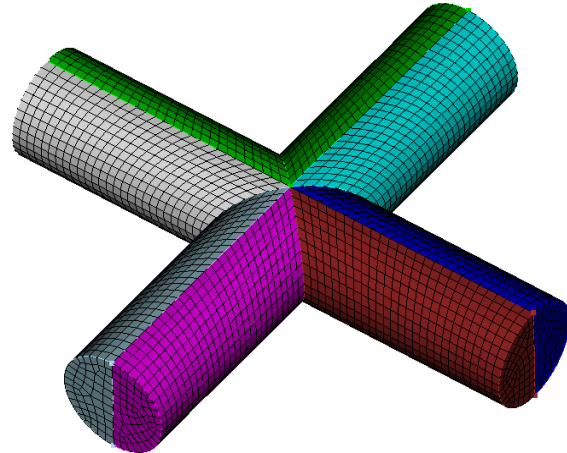
```

reset
create cylinder height 20 radius 2
volume 1 copy rotate 90 about y
unite body all

webcut volume all with loop curve 5 7
webcut volume all with loop curve 8 6
merge all

volume 5 scheme Sweep source surface 16 17 target surface 3
volume 4 scheme Sweep source surface 13 16 target surface 6
volume 3 scheme Sweep source surface 14 17 target surface 5
volume 1 scheme Sweep source surface 13 14 target surface 2
mesh vol all
  
```

Decompose Only



```

reset
create cylinder height 20 radius 2
volume 1 copy rotate 90 about y
unite body all

webcut volume all with loop curve 5 7
webcut volume all with loop curve 8 6
merge all
webcut volume all with loop curve 5 7
webcut volume all with loop curve 8 6
webcut volume all with plane xplane
webcut volume all with plane zplane
mesh vol all
  
```

sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

I would not go into details, but there are two ways to do it.

The user's only job is to do one or both of the following

1. For volumes that cannot be meshed, specify a scheme that can be meshed.
2. Decompose a Many-to-Many type volume into Many-to-One, or One-to-One, so that CUBIT can automatically select a Meshable scheme.

The most important thing for “Mesh Vol ALL”

- CUBIT provides the **tools** to do the hexa meshing that the user imagines.
 - Meshing scheme
 - **Surface : Map, Submap, Pave, etc.**
 - **Volume : Map, Submap, Sweep, etc.**
 - Geometry decomposition tools
 - **Webcut**
- **The main task** of the user is to **decompose the model to be simulated** (typically a Many-to-Many volume) **from Many-to-Many to Many-to-One/One-to-One** in order to achieve the hex meshing that the user imagines.



sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

Here is a summary of the important things to know about using the semi-automatic hex meshing feature in “Mesh Vol All”.

CUBIT is only a tool for the user to create the hex meshing they have imagined.

Also, the user's main task is only to decompose the Many-to-Many volume into Many-to-One or One-to-One. Thinking about this is not the job of CUBIT, but of the user.

The wonderful “ALL” and Cubit journal systems

For example, some of our customers may have at most 10 Volumes. In such a case, it is hard to understand the advantages of CUBIT.

Suppose the number of Volumes is 100.

Let's say it takes 3 minutes to create a mesh for one volume, specifying the mesh size and the mesh scheme. Then it would take 300 minutes to mesh 100 volumes. In other words, 5 hours.

Well, this is an acceptable amount of time, isn't it? If your boss asks you to change the mesh distribution, it will also take 5 hours to create the mesh again.

CFD users sometimes have volumes of about 2000 to 3000. 3000 volumes is 9000 minutes, or 150 hours, or 30 days if you spend 5 hours a day meshing. That's a very high labor cost.

Time required to set up and create the hex mesh and labor costs (Assume 5 hours of work per day and an hourly wage of \$50 per hour.)				
Number of Volumes	Minutes	Hours	Days	Cost \$
1	3	0.05	1	2.5
10	30	0.5	1	25
100	300	5	1	250
1000	3000	50	10	2,500
3000	9000	150	30	7,500
9000	27000	450	90	22,500



If I search "US CAE engineer fee" on Google, I get "average total cash compensation of \$86,584 ". If you also search for "US working hours ", you will find "1815 hours ". The unit price per hour is about \$50.

coreform.com
yosun@coreform.com

I will say it again about the greatness of All and CUBIT's journal system.

As I have just described, CUBIT is

1. Intuitive geometry creation
2. Semi-automatic mesh generation by simply sending command to CUBIT “mesh vol all”
3. Using the journal file, it is very easy to change the mesh size and generation method.

Maybe you are wondering, "Is it true?"

You will soon find out how great All is.

The wonderful "ALL" and Cubit journal systems

For example, some of our customers may have at most 10 Volumes. In such a case, it is hard to understand the advantages of CUBIT.

Suppose the number of Volumes is 100.

Let's say it takes 3 minutes to create a mesh for one volume, specifying the mesh size and the mesh scheme. Then it would take 300 minutes to mesh 100 volumes. In other words, 5 hours.

Well, this is an acceptable amount of time, isn't it? If your boss asks you to change the mesh distribution, it will also take 5 hours to create the mesh again.

CFD users sometimes have volumes of about 2000 to 3000. 3000 volumes is 9000 minutes, or 150 hours, or 30 days if you spend 5 hours a day meshing. That's a very high labor cost.

Time required to set up and create the hex mesh and labor costs (Assume 5 hours of work per day and an hourly wage of \$50 per hour.)				
Number of Volumes	Minutes	Hours	Days	Cost \$
1	3	0.05	1	2.5
10	30	0.5	1	25
100	300	5	1	250
1000	3000	50	10	2,500
3000	9000	150	30	7,500
9000	27000	450	90	22,500



If I search "US CAE engineer fee" on Google, I get "average total cash compensation of \$86,584 ". If you also search for "US working hours ", you will find "1815 hours ". The unit price per hour is about \$50.

coreform.com

yosimeta.tokuyama@jpmandt.com

For example, some of our customers may have at most 10 Volumes. In such a case, it is hard to understand the advantages of CUBIT.

Suppose the number of Volumes is 100.

Let's say it takes 3 minutes to create a mesh for one volume, specifying the mesh size and the mesh scheme. Then it would take 300 minutes to mesh 100 volumes. In other words, 5 hours.

Well, this is an acceptable amount of time, isn't it? If your boss asks you to change the mesh distribution, it will also take 5 hours to create the mesh again.

The wonderful “ALL” and Cubit journal systems

For example, some of our customers may have at most 10 Volumes. In such a case, it is hard to understand the advantages of CUBIT.

Suppose the number of Volumes is 100.

Let's say it takes 3 minutes to create a mesh for one volume, specifying the mesh size and the mesh scheme. Then it would take 300 minutes to mesh 100 volumes. In other words, 5 hours.

Well, this is an acceptable amount of time, isn't it? If your boss asks you to change the mesh distribution, it will also take 5 hours to create the mesh again.

CFD users sometimes have volumes of about 2000 to 3000. 3000 volumes is 9000 minutes, or 150 hours, or 30 days if you spend 5 hours a day meshing. That's a very high labor cost.

Time required to set up and create the hex mesh and labor costs (Assume 5 hours of work per day and an hourly wage of \$50 per hour.)				
Number of Volumes	Minutes	Hours	Days	Cost \$
1	3	0.05	1	2.5
10	30	0.5	1	25
100	300	5	1	250
1000	3000	50	10	2,500
3000	9000	150	30	7,500
9000	27000	450	90	22,500



If I search "US CAE engineer fee" on Google, I get "average total cash compensation of \$86,584 ". If you also search for "US working hours ", you will find "1815 hours ". The unit price per hour is about \$50.

coreform.com
yosimura@coreform.com
yosimura@coreform.com
yosimura@coreform.com

CFD users sometimes have volumes of about 2000 to 3000. 3000 volumes is 9000 minutes, or 150 hours, or 30 days if you spend 5 hours a day meshing. That is a very high labor cost.

If I search “US CAE engineer fee” on Google, I get “average total cash compensation of \$86,584”. If you also search for “US working hour”, you will find “1815 hours”. The unit price per hour is about \$50.

If you do 150 hours, the cost is \$7,500.

If you can create a volume mesh all in CUBIT, it will only cost you \$50!

The wonderful “All” and Cubit journal systems

Whether you have 1 volume, 10 volumes, 100 volumes, 1000 volumes, or 10000 volumes, you can create the mesh by specifying ALL!

Is CUBIT an expensive product?

If you have a labor cost of \$1000 per day, you can make one mesh with Cubit and you got your money's worth.



sales@coreform.com
yoshiteru.tokuyama@jpmandt.com

Whether you have 1 volume, 10 volumes, 100 volumes, 1000 volumes, or 10000 volumes, you can create the mesh by specifying ALL!

Is CUBIT an expensive product?

If you have a labor cost of \$1000 per day, you can make one mesh with CUBIT and you got your money's worth.