Title:	**Developments in CFD Responding to the Increased Demands of Taller Buildings**
Authors:	Darren Davies, Technical Director, Wirth Research Jon Winchester, Chief CFD Engineer, Wirth Research
Subject:	Wind Engineering
Keywords:	Aerodynamics CFD Wind Wind Tunnel Testing
Publication Date:	2014
Original Publication:	CTBUH 2014 Shanghai Conference Proceedings
Paper Type:	1. Book chapter/Part chapter 2. Journal paper 3. **Conference proceeding** 4. Unpublished conference paper 5. Magazine article 6. Unpublished

Developments in CFD Responding to the Increased Demands of Taller Buildings
计算流体力学的发展与其在高层建筑中与日俱增的需求

Darren Davies

Jon Winchester

Darren Davies & Jon Winchester

Wirth Research Limited
Unit D4 Telford Road
Bicester, Oxfordshire
OX26 4LD United Kingdom

tel (电话): +44 1869 355260, +44 1869 355260
fax (传真): -
email (电子邮箱): darren.davies@wirthresearch.com; jon.winchester@wirthresearch.com
www.wirthresearch.com

Darren Davies has been heading the Technical development of Wirth Research since 2006. As Technical Director, Darren has responsibility for all engineering projects at Wirth Research Limited, (WR) encompassing Design, CFD, Aerodynamics, Software design, Simulation, R&D, Motorsport, Architectural and Commercial vehicle engineering. Darren directs the company's development in all aspects of its advanced and virtual engineering and has been responsible for WR's activities within the Architectural sector since 2008 with a specialised focus on the design and performance of Tall Buildings and minimising their environmental impact through better informed design.

Darren从2006年起开始领导维尔特研究有限公司 (WR)的科技研发。 作为技术总监，Darren负责的工程项目包括: 设计、计算流体力学、空气动力学、软件设计、模拟、研发、赛车、建筑工程和商业船舶工程。 Darren全方位领导公司在高端和虚拟工程领域的项目。从2008年起，他开始负责WR公司在建筑领域的工程，专注于超高层建筑设计和性能模拟，从而减少建筑对环境的影响。

Jon Winchester is the Chief CFD Engineer at Wirth Research, with responsibility for development and maintenance of CFD methods and tools across all aerodynamic projects within the company. Jon has a leading role developing the growth of Wirth Research's high resolution architectural CFD capability, and has worked on projects concerned with tall buildings, master planning, pedestrian comfort and natural ventilation with a range of International architects including Foster & Partners.

Jon Winchester是WR公司的首席计算流体力学工程师，负责计算流体力学软件的研发和维护，设计公司所有的空气动力学项目。 Jon 领导WR公司的高精度建筑计算流体力学领域的研发，他在超高层建筑、城市规划、行人舒适性、自然通风领域与许多国际知名的建筑师们有合作，其中包括与福斯特建筑事务所 (Foster & Partners) 的合作。

Abstract

Accurate analysis of the aerodynamic effect of tall buildings is vital for the sustainable development of the vertical urban habitat. The established analysis methods are wind tunnels and low-resolution CFD. Wind tunnels are unable to operate at anything approaching full scale, which inhibits their ability to re-create the correct Reynolds number regime and capture boundary layer driven separation. Low resolution CFD cannot capture all of the relevant details which may cause geometry driven separation. High resolution CFD takes advantage of recent advantages in super-computing to accurately model realistic flow conditions at full scale. It can mimic wind tunnel results at small scale, whilst also providing the ability to go far beyond the limits of wind tunnels without any loss of fidelity. When coupled with world-class aerodynamic expertise, it provides better information, to inform faster, higher quality decisions at all stages of the process.

Keywords: High-Resolution Architectural CFD, Building Aerodynamics, Flow Separation, Wirth Research

摘要

精确的超高层建筑空气动力学分析对城市纵向居住环境的可持续拓展有极其重要的作用。现有的分析方法包括风洞和低精度的计算流体力学模拟。对于大型建筑，风洞模型不适合在原始大尺寸下运行，从而影响它捕捉正确的雷诺数和边界层的能力。低精度计算流体力学模拟不能捕捉到全部细节，从而导致几何属性驱使的流体分离。 高精度流体力学模拟利用尖端的大型计算机可以逼真的模拟大空间的气流。它可以仿真小型风洞的结果，并且有能力超越风洞的限制而且不损失逼真度。

关键词: 高精度建筑流体力学，建筑气动学，流动分离，维尔特研究

Tall Building Aerodynamics with High Resolution CFD

Between 2000 and 2013 the global number of tall buildings (over 200m) increased from 261 to 829. Since 2010 at least 69 new tall buildings have been added each year (CTBUH 2014). For buildings of this size, their aerodynamic response to high winds is of paramount importance, as the unstable forces caused by vortex shedding can be substantial. Tall buildings can also significantly accelerate the wind speeds near ground level, and cause discomfort to pedestrians, which needs to be mitigated. Typically, the options available for aerodynamic analysis of tall buildings are wind tunnels and standard-resolution Computational Fluid Dynamics (CFD).

Wind tunnels have the advantage of being an established technology, with practices and techniques developed over many years of experience. They are, however, limited by size and can only operate at small scale. For instance, even at 1/250 scale, a model of Burj Khalifa (832m) will be over 3.3m tall.

超高层建筑气动学与高精度计算流体力学

从2000年到2013年，世界超高层建筑 (200米以上) 的数量从261座增加到829座。从2010年开始，每年至少新增69座超高层建筑 (CTBUH 2014)。对于超高层建筑，它们的气动学设计对高速风的反应至关重要，因为不稳定风力造成的涡旋很有严重的后果。超高层建筑也会是显著增加地表的风速，导致行人的不舒适 (应该避免)。通常，超高层建筑的气动学分析方法有风洞分析和一般精度的计算流体力学分析。

风洞是一种成熟的技术，人们已经有很多年的使用经验。但是，它有尺寸的限制，只能模拟按比例缩小的模型，比如，1/250比例的迪拜哈利法塔仍然超过3.3米高 (原高度832米)。

很多标准精度的计算流体力学模拟一般可以在台式计算机上进行，大约两千万个网格的计算需要数天来进行。在这种精度条件下，总要平衡精度与计算速度。因此，气动学模拟不常在建筑设计过程中使用。

最新的高性能计算机为计算流体力学模拟提供了潜力，高精度计算流体力学模拟现在已经是一个实际可行的选项。高精度计

Most standard-resolution CFD is typically performed on a desktop PC, and will be able to find solutions for meshes of approximately 20 million cells in a matter of days. At this resolution, there is a constant trade-off between accuracy and speed, and subsequently this tool falls short of the level required for consistent use throughout the design process.

Recent advances in super-computing have unlocked the potential of CFD, and high-resolution CFD is now a realistic option. High-resolution CFD uses parallel processing across many cores of a High-Performance Computing (HPC) system. With high-resolution CFD it is possible to solve meshes of over 1 billion cells in a matter of hours. This paper is concerned with the potential for use of high-resolution CFD in the design of tall buildings.

The primary feature of the aerodynamic behavior of a tall building is the point at which the flow around the building separates and a wake is formed. Flow separation can be boundary layer driven, or geometry driven. For an aerodynamic tool to be truly useful in building design, it needs to be able to accurately capture both of these phenomena.

Boundary Layer Driven Separation

Cylinder Flow. One of the most conceptually simple, yet physically complicated, flow conditions to model is that of the flow around a circular cylinder. As a cylinder does not have any edges, to "trip" the flow, any separation that occurs is entirely boundary layer driven. How a boundary layer is expected to behave is determined by the Reynolds number, Re, which is the ratio of inertial forces to viscous forces within the boundary layer.

$$Re = \rho Vc/\mu$$

Where ρ is the fluid density, V is the freestream fluid velocity, c is a characteristic length (the diameter, in the case of a circular cylinder) and μ is the dynamic viscosity of the fluid. The flow around a cylinder can be described by 4 distinct regimes, "sub-critical", "critical", "super-critical" and "hyper-critical" (also known as "trans-critical"). These regimes, as well as a review of wind tunnel experiments on circular cylinders, were described by Polhamus (Polhamus 1984).

For "sub-critical" flow (Re < 2x105), the flow within the boundary layer is entirely laminar. In this case the flow separates early (approximately 80°), which results in high cylinder drag and vortex shedding with a distinct, dominant period.

For "critical" flow (2x105 < Re < 4x105), the boundary layer experiences transition from laminar to turbulent flow. This regime is characterized by the formation of a "laminar separation bubble", where the flow transitions to turbulent after the laminar separation point, and briefly re-attaches prior to the separation point at approximately 130°. During this regime the drag reduces significantly, and the vortex shedding loses its dominant periodicity to become random "wide-band". In this regime, the flow is highly sensitive to Reynolds number.

For "super-critical" flow (4x105 < Re < 6x106), the boundary layer is entirely turbulent. Initially, the cylinder drag is low and the vortex shedding is "wide-band" random. The separation point moves upstream to approximately 115° and the drag rises steadily. At the upper end of this regime (Re > 3.5x106), the drag levels out and the vortex shedding stabilizes to "narrow-band" random.

"Hyper-critical" flows (Re > 6x106) are beyond the scope of most conventional wind tunnels, but can be investigated experimentally using different fluids, such as Freon. In this regime, the random vortex

算流体力学模拟运用高性能计算机的并行处理技术。它可以在几小时内计算超过十亿的网格。这篇文章研究高精度计算流体力学模拟在超高层建筑设计中的运用。

超高层建筑的气动学问题主要集中在环绕建筑外部的气流分离和其形成的尾流。气流分离可以是边界层驱使的或者是几何属性驱使的。对于真正用于超高层建筑设计气动学研究的工具，它应该能够准确的捕捉这两点。

边界层驱使的气流分离

圆筒流动: 一种概念最简单却物理上复杂的流动条件——气流环绕着圆柱体流动。因为圆柱体没有任何棱角，任何发生的分离完全由边界层所驱使。边界层的状态是由雷诺数 (Re) 来决定的。雷诺数表征流体受惯性力与粘性力之比。

$$Re = \rho Vc/\mu$$

ρ，流体密度; V，自由流体速度; c，长度特征尺度 (圆柱体的直径) ; μ，流体粘度。气流环绕着圆柱体流动可以本描述为4种不同的状态: 亚临界，临界，超临界 (super-critical) 和跨临界 (hyper-critical，也叫trans-critical) 。这些状态被描述在Polhamus (1984) 的文献中，他的文章中也总结了圆筒流动的风洞试验结果。

亚临界流 (Re < 2x105) ， 边界层里流体完全是层流。在这种情况下，流体提早分离 (大约80度) 导致相当一段时间内的强大空气阻力和旋涡。

临界流 (2x105 < Re < 4x105) ， 边界层经历从层流到紊流的转变过程。这个状态被描述为"层状分离气泡"，此时流体超过层流分离点变位紊流，分离点大约快到达130度。在这个状态中，空气阻力限制减小，旋涡分离损失的显著周期性变为随机"宽频"。这种状态下，流体和雷诺数直接相关。

超临界流 (4x105 < Re < 6x106) ， 边界层完全紊流。空气阻力低，旋涡分离"宽频"随机。分离点使逆流到大约115度，空气阻力稳定增加。在这个状态的顶部 (Re > 3.5x106) ，空气阻力达到平衡，旋涡分离稳定在"窄频"随机。

跨临界流 (Re > 6x106) ， 超过了一般风洞范围，但是可以用不同的流体去试验，比如氟利昂。在这种情况下，临界和超临界状态下的随机旋涡分离被准周期旋涡分离所取代，空气阻力减小。

大多数超高层建筑处在超临界状态。比如一个圆柱形建筑 (新加坡aka Axa 塔，Shenton路8号) 被一个50米直径的圆柱体所代表，那么可以相对可靠的假设200米高处风速10m/s，空气密度1.15kg/m3 、空气粘度1.8x10-5 kg/m s。雷诺数3.2x107。这完全超越了传统风洞的规模。考虑到可靠的大气层流的重要性，这几乎不可能用全尺寸风洞去正确捕捉超高层建筑的边界层情况。

高精度计算流体力学圆筒模型: 为了研究高雷诺数的圆筒模型以及准确捕捉边界层分离的需求，作者进行一个基于圆筒风洞数据 (Jones et al 1969) 的验证研究。圆筒处在超临界的底部，雷诺数8.4x106，气动平稳。

因为旋涡分离的频率是一个重要的研究点，模拟由附带漩涡模型 (Detached Eddy Simulation, DES) 运行。DES是个高阶紊流模型，它可以在尾流区域准确的模拟紊流结构。CFD模拟采用三层网格: 疏散 (4千万网格) ，中等 (6千万网格) ，精密 (8千万网格) 。

图1显示圆柱表面的压力分布，我们可以看出最小压力的强度和位置 (大约80度) 和网格大小有很紧密的关系。从此可以看出，为了准确捕捉这些因素，至少中等精度的网格是需要的。尾流的结构研究发现 (见图2) ，尽管使用了中等大小的网格，只有很少的尾流结构能被捕捉到 (相比较与精密网格) 。这些尾流结构对于

shedding of the "critical" and "super-critical" regimes is replaced by "quasi-periodic" vortex shedding, and the drag is reduced.

The majority of tall buildings will be within the "hyper-critical" regime. For instance, if a cylindrical building (such as 8 Shenton Way, aka Axa Tower, Singapore) is represented by a cylinder of 50m diameter, then it could reasonably be assumed that it will be regularly subject to wind speeds of ~10m/s at 200m height, where the density of the air could typically be ~1.15kg/m^3 and the dynamic viscosity ~1.8x10-5 kg/m s. This would have a Reynolds number of 3.2x107, which is far beyond the scope of a traditional wind tunnel. Combined with the importance of creating a realistic Atmospheric Boundary Layer (ABL), it is highly unlikely to be possible for a wind tunnel to capture the correct boundary layer flow regimes experienced by a tall building at full scale.

High Resolution CFD of a Cylinder. To investigate the requirements for accurate capture of boundary layer separation around a high Reynolds number cylinder, a validation study was performed against published wind tunnel data for a cylinder at the lower end of the "hyper-critical" range (Jones et al 1969). The cylinder used had Reynolds number of 8.4x106 and was aerodynamically smooth.

As vortex shedding frequency was a key point for investigation, the simulations were run transient using Detached Eddy Simulation (DES). DES is a higher order turbulence model, which allows accurate modeling of the turbulent structures in wake regions. The CFD was run with 3 levels of mesh; coarse (40 million cells), medium (60 million cells) and fine (80 million cells). It should be noted that even though it is referred to as coarse here, the "coarse" run is of significantly greater resolution than could be solved in a timely manner by most standard resolution CFD (typically limited to 20 million cells).

When looking at the pressure distribution on the surface of the cylinder (see Figure 1), it is apparent that the magnitude and location of the minimum pressure (at approximately 80°) is sensitive to mesh size. From this it is apparent that at least the medium mesh is necessary in order to capture these factors accurately. From analysis of the wake structures (see Figure 2), it is apparent that, even if the medium mesh is used, fewer flow structures are captured than by the fine mesh case (the importance of this will be magnified significantly if there are multiple buildings downstream and upstream of the building of interest). This demonstrates a direct correlation between the mesh grade, and the quality of information generated, which will facilitate better design decisions.

Geometry Driven Separation

The CAARC Standard Tall Building Model. In many cases, flow separation is not driven by the boundary layer, and is instead caused by specific geometric features such as sharp edges or ridges. In the presence of features of this type; a sudden change in the shape of an aerodynamic surface can cause an adverse pressure gradient, which leads to flow separation.

A good test case for a building which is likely to experience geometry driven separation is the Commonwealth Advisory Aeronautical Research Council (CAARC) standard tall building model. The CAARC model is a rectangular cuboid (a typical profile of many tall buildings) measuring 30.48m x 45.72m in plan, and of height 182.88m. It was proposed in 1969 as a standard model which could be used as a test case for different wind tunnels and modeling techniques. Consequently there is a large body of test data which can be drawn upon regarding this building.

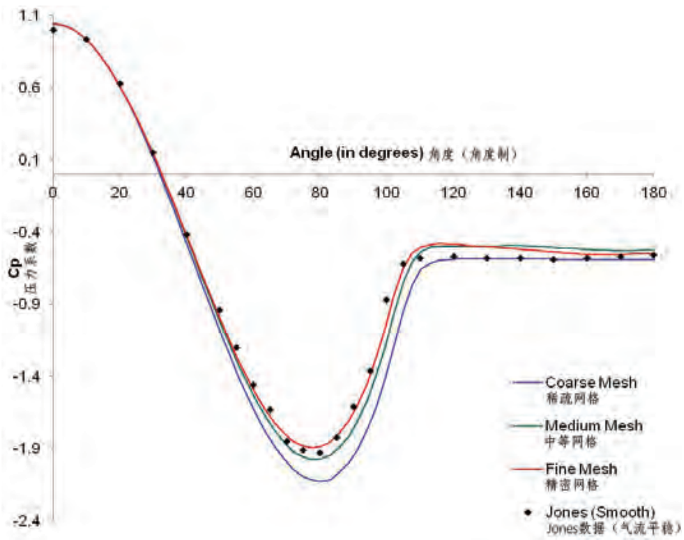

Figure 1. Measured pressure coefficient values for mid plane of a circular cylinder.
图1. 测量的圆柱体中间界面的压力系数

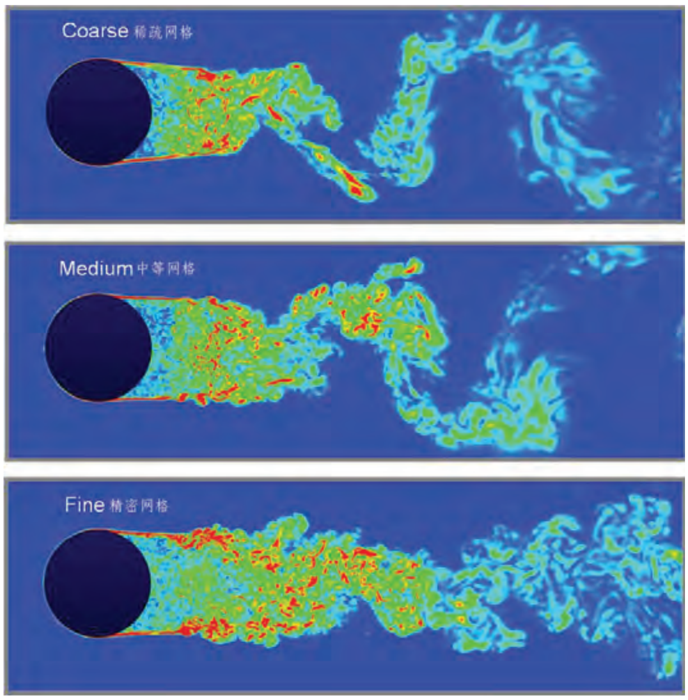

Figure 2. Vortex shedding in the wake of a circular cylinder.
图2. 圆柱体所产生的尾流涡旋

建筑群中处在风头和风尾的建筑至关重要。这个研究展示了网格精度和其所能产生信息质量的关系，也能够帮助做更好的设计决策。

几何属性驱使的气流分离

英联邦航空顾问研究委员会 (CAARC) 高层建筑模型: 在很多条件下，流体分离不是由边界层所驱使的，而是由特定的几何特性所导致的，比如尖利的边缘或者脊梁。这种现象的表现有: 突然的气动表面变化会导致压力的反向变化，因此导致流体分离。

英联邦航空顾问研究委员会 (CAARC) 标准高层建筑模型是一个很好的几何属性驱使的气流分离案例。CAARC模型是一个长45.72米、宽30.48米、高182.88米的长方体 (很多高层建筑的典型外形)。1969年，它被建议为标准模型用于测试不同的风洞和模拟软件。因此，这座建筑有大量的测试数据可用。

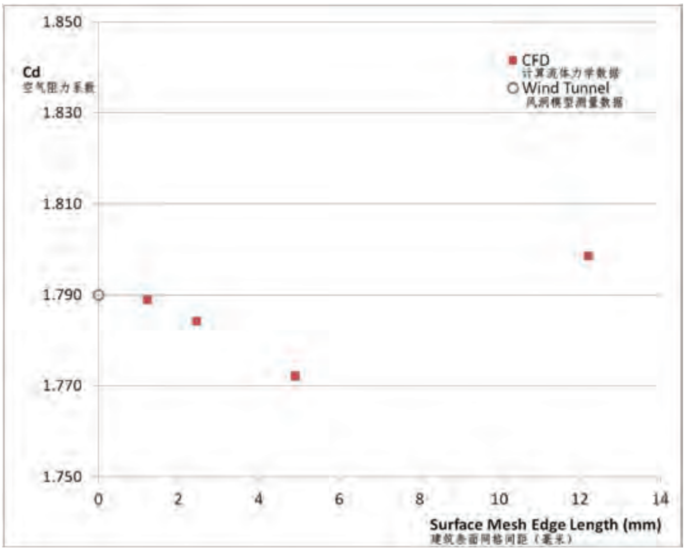

Figure 3. Measured drag coefficient values for 1/250 scale CAARC standard tall building.
图3. 测量的1/250 CAARC标准超高层建筑的空气阻力系数

The majority of comparable wind tunnel data is available at between 1/250 and 1/1000 scale, although some of these do not have comparable turbulent length scales for the building scale (Obasaju 1992).

High Resolution CFD of CAARC Standard Tall Building Model. To investigate the requirements for accurate prediction of geometry driven separation, high resolution CFD was performed of the CAARC standard tall building model at 1/250 scale.

The data source used (Obasaju 1992) was chosen due to its relatively large scale (1/250), which is well matched by the turbulent length scales measured. This data set is mainly concerned with time-averaged drag values, so Reynolds-Averaged Navier-Stokes (RANS) was the main tool used (RANS provides a steady state solution, with transient fluctuations smoothed out). To highlight any features which RANS (which is the only viable tool in most standard resolution CFD) may miss, a DES run of the same geometry was also performed.

The 1/250 scale CFD used 4 meshes. The finest of these, "fine", had an evenly distributed mesh of 1.225mm edge length on the building surfaces (~300mm at full scale, which is the recommended edge length for use in high resolution architectural CFD). This volume mesh consisted of 73 million cells. The "medium" mesh had double the edge length (2.45mm) and consisted of 12 million cells, and the "coarse" mesh had double the edge length again (4.9mm) and consisted of 2 million cells. The fourth mesh was created in accordance with the guidelines of the European Co-Operation in Science and Technology (COST) Action 732 (Franke et al 2007), which states that 10 cells per building side is necessary. As the minimum dimension of the 1/250 scale CAARC model is 122mm, this equates to 12.2m edge length, which results in a volume mesh of 0.3 million cells (known from here as the COST mesh).

The drag results (see Figure 3), show that if the fine mesh is used, the drag coefficient is within 0.001 of the experimental results. The Cd error is 0.006 for the medium mesh and 0.018 for the coarse mesh (at full scale, in 10m/s wind, this equates to an error of ~250N for the fine mesh ~2.4kN for the medium mesh and ~9kN for the coarse mesh). The COST mesh is closer to the experimental result than the coarse mesh result. This should be treated with caution, however, as it is

对建筑来说，尽管一些风洞数据没有可比较的紊流长度尺寸，主要的风洞数据是基于1/250和1/1000的比例尺寸 (Obasaju 1992) 。

高精度CFD模拟CAARC标准建筑模型: 为了研究精确预测几何属性驱使的气流分离所需的条件，高精度的CAARC标准建筑CFD模型采用1/250比例。

数据源 (Obasaju 1992) 被选中是因为1/250是相对较大的比例尺寸，它能很好的与测量的紊流长度尺寸相匹配。这些数据侧重于时间平均的空气阻力，所以雷诺平均纳维斯托克斯方程 (Reynolds Averaged Navier-Stokes, RANS) 是主要的工具。雷诺平均纳维斯托克斯方程 (RANS) 提供静态解决方案，瞬态的波动被平滑掉了。为了强调RANS方法所产生的缺失 (RANS被很多标准CFD工具所采用) ，附带漩涡模型 (DES) 被用在同样的几何模型上。

1/250比例的CFD模型采用四种网格。精细网格，在建筑表面采用1.225毫米的均匀网格 (高精度建筑CFD模拟所推荐的网格在300毫米左右) 。这样的模型网格数在七千三百万。中等密度的网格采用2.45毫米密度，它有一千二百万个网格。稀疏密度的网格采用4.9毫米的间距，它有两百万个网格。第四种网格采取欧洲科技协作732条 (European Co-Operation in Science and Technology (COST) Action 732， Franke et al 2007) 所建议的: 在需要的情况下，每面个建筑表面采用十个网格。因为1/250比例 CAARC标准建筑模型的最小单位是122毫米，这等同于12.2毫米的网格间距，这种情况下的模型有三十万的网格 (简称COST网格) 。

空气阻力的结果显示 (见图3) : 和风洞模型的相比，精细网格的阻力系数与实测结果相差小于0.001。中等密度模型的阻力偏差是0.006，稀疏模型的偏差是0.018。在全尺寸10m/s风速的模型下，这等同于大约250牛 (精细网格) 、2400牛 (中等网格) 、9000牛 (稀疏网格) 的偏差。COST网格比稀疏网格更接近实测数据。需要注意的是，任何比中等密度网格更稀疏的网格对网格的大小有很高的依赖度，这对模型的可重复性有很重要的影响。

附带漩涡模型 (DES) 的解决方案可以捕捉到雷诺平均纳维斯托克斯 (RANS) 方法不能捕捉的细节 (见图4) 。彩色的压力图像显示一个区域有持续的Lambda-2 条件，这是绘制涡旋的标准办法。尽管与建筑紧邻的尾流区的形状和压力分布大致相似，有很多细小的涡旋可以被DES模型所捕捉，却被RANS模型忽略。这在下风口处更明显。对于几何模型所驱使的流体分离，更细的网格，更高阶的涡旋模型可以显著的提供高质量、有实际意义的信息。

案例: Trump国际酒店塔楼

在实际情况下，超高层建筑的气流分离是边界层驱使和几何模型驱使的结合。芝加哥的Trump国际酒店塔楼 (以下简称Trump塔楼) 被用于案例分析。它是由SOM建筑事务所 (Skidmore, Owings & Merrill LLP) 设计，2009年封顶的。Trump塔楼被选择作为案例分析是因为它的设计包含了边界层驱使或者几何模型驱使的因素，所以它可以避免累加的涡流，从而减少建筑结构的风负荷。这使这个415米的塔楼没有采用风振控制装置 (Baker et al 2009) 。

全尺寸的Trump塔楼采用RANS模拟。与CAARC模拟类似，三种不同的网格被采用: 精密网格、中等网格、稀疏网格。相应的网格总数是1.6亿 (精密) 、2800万 (中等) 、600万 (稀疏) 。

显而易见的是，当建筑外形比圆柱体或者长方体更复杂的时候，稀疏网格模型会导致失掉很多细节。丢失掉的细节 (见图5) 会导致几何外形引起的气流分离，因此越详细的建筑模型，网格的大小越敏感。

从视觉上检查Trump塔楼的尾流涡旋 (见图6) ，“混合”的气流分离

apparent that for anything coarser than the medium mesh the solution is highly mesh dependant, which could have serious implications for the repeatability of any options run using a mesh of this resolution.

Running the solution in DES shows the transient flow features which cannot be captured by RANS (see Figure 4). These images show regions of constant Lambda-2 criterion, which is a standard method of vortex visualization, colored by total pressure. While the general shape and pressure distribution of the wake in the immediate vicinity of the building can be seen to be broadly similar, there are numerous smaller vortical structures captured by the DES, which are smoothed out by the RANS. This is particularly noticeable further downstream. As with boundary layer driven separation, using the finer mesh and higher order turbulence modeling, results in considerably higher quality, actionable, information.

Trump International Hotel and Tower: Case Study

In reality, the flow separation around most tall buildings is a combination of boundary layer driven and geometry driven. The Trump International Hotel and Tower in Chicago (henceforth referred to as Trump tower), which was designed by Skidmore, Owings & Merrill, LLP (SOM) and topped out in 2009, is used as a case study. The Trump tower is of particular interest, as it was designed specifically with the intention of mixing up whether the separation is boundary layer driven or geometry driven, so as to prevent the build-up of wind vortices and reduce the unstable wind loads experienced by the structure. This has enabled the 415m tower to be built without needing dampers (Baker et al 2009).

RANS simulations were performed of the Trump tower (at full scale) with fine, medium and coarse mesh created to the same specification as used for the CAARC simulations (0.3m fine, 0.6m medium, 1.2m coarse). The corresponding volume mesh sizes are 160million cells (fine), 28 million cells (medium) or 6 million cells (coarse).

Of immediate note is that, when meshing a building of greater detail than a circular cylinder or a rectangular cuboid, a coarse mesh can lead to significant de-featuring. The features which are removed (see Figure 5) could be instrumental in causing geometry induced separation, and consequently the greater the detail of the building, the greater the sensitivity of the solution to mesh size.

From visual examination of the vortices in the wake of the Trump tower (see Figure 6), it is apparent that the "mixed-up" flow separation (as intended by the design) can be seen for all three meshes. What is also apparent is that the vortical structures being shed from the tower break up noticeably earlier for the coarse mesh case (note that the coarse mesh case is still considerably finer than recommended by COST 732). Although the medium mesh and the fine mesh appear similar, the medium mesh over-predicts drag by 5.5% compared to the fine mesh, which is likely due to early prediction of the boundary layer separation in this case. This again demonstrates that high resolution CFD can provide information that is not available via traditional, standard resolution methods.

It should be noted that, although this is analysis of an existing building, these tools can be used throughout the design process, resulting in the ability to access better information, faster and to make better design decisions, earlier.

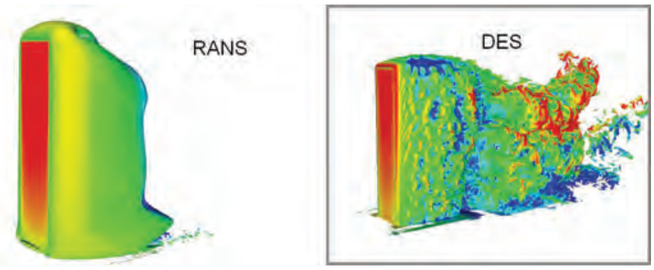

Figure 4. Comparison of CAARC standard tall building wake according to RANS or DES simulations.
图4. CAARC标准超高层建筑气流尾流比较 (RANS和DES模拟)

Figure 5. Geometry of Trump tower for coarse and fine mesh simulations.
图5. Trump塔楼的稀疏网格和精密网格

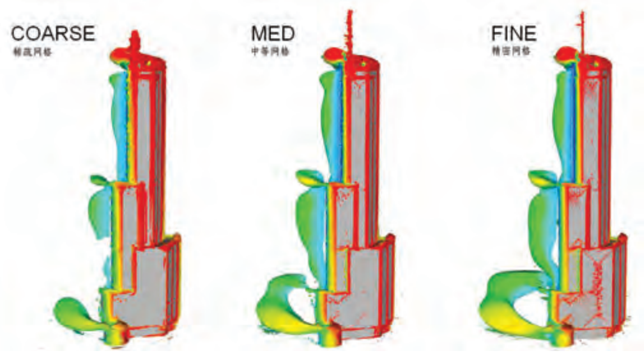

Figure 6. Vortex shedding from the Trump tower.
图6. Trump塔楼所产生的气流涡旋

(设计所追求的) 可以在三中网格中都被发现。我们也可以发现，塔楼所引起的涡旋结构在稀疏模型中明显更早地被消散 (稀疏模型仍然显著高于COST 732所建议的网格精度)。尽管中等密度网格和精细网格结果相似，中等密度网格过大的预测了空气阻力，它比精细网格高5.5%。这很可能由于早期预测的边界层分离。这个案例同样显示，高精度的CFD模拟可以提供传统、标准模拟所不能提供的信息。

需要提醒的是，尽管这些分析是基于现有建筑的，这些工具是可以用于设计的全过程，帮助提供更好信息，更快更早更好的用于设计决策。

城市规模模拟

高精度的单体建筑CFD模拟本身是一个非常有用的工具，它可以用于早期气动学的分析，从根本上指导建筑设计的过程。更强大的功能是，这个工具有能力建立细节的模型去分析建筑及其周边环境。在不损失建筑网格的情况下，有可能把建筑放在一个真实的本地化的风环境中。满足这种条件的CFD模型 (至少上亿的网格，有可能超过十亿) 需要高级复杂的高性能计算机系统，比如维尔特研究公司 (WR) 所拥有的设备。

把建筑放在一个真实的城市环境中，不仅仅去研究建筑对周围环

Cityscapes

While the high resolution CFD study of single buildings in isolation can be a useful tool for early stage aerodynamic analysis, which can inform the design process at a fundamental level, the greatest power that this tool has is the ability to place a building into a detailed model of the surrounding area. Without reducing the resolution of the mesh on the building of interest, it is possible to place it within realistic local wind conditions. It is possible to meet the mesh requirements of this level of CFD (hundreds of millions of cells minimum, potentially over a billion cells) only by use of a highly sophisticated HPC system, such as that belonging to Wirth Research.

By placing a building in a realistic cityscape, it is not only possible to assess the aerodynamic impact of the building on its surroundings (and of the surroundings on the building), but also to compare the aerodynamic behavior of the building of interest to other local tall buildings. For instance, if the Trump tower is simulated within a 10km diameter of the rest of Chicago (see Figure 7), there is a distinct contrast between the large, unsteady, sheet vortices shed by the Willis Tower and the John Hancock Center, and the smaller, distinct vortices shed by the Trump tower.

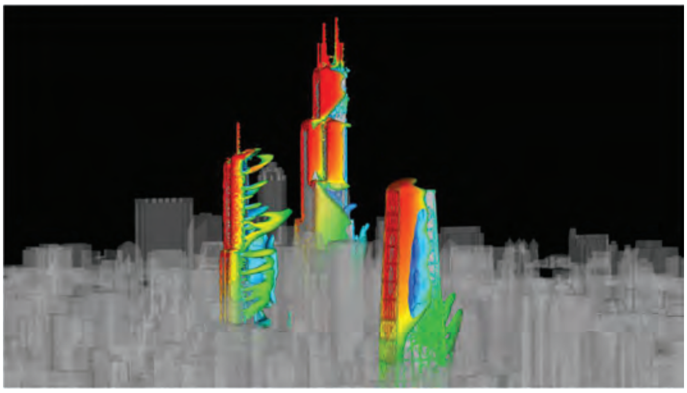

Figure 7. Vortex shedding from the Trump tower, Willis tower and John Hancock center, in realistic 10km Chicago cityscape.
图7. 芝加哥10公里城市模型， 显示Trump塔楼、Willis塔楼和John Hancock中心所产生的气流涡旋

境的气动学影响 (和周围环境对建筑的气动学影响)，也包括比较这个建筑和附近其他高层建筑的气动学行为。比如，如果模拟芝加哥Trump塔楼方圆10公里的范围 (图7)，我们会很明显的对比发现，Willis 塔楼 和 John Hancock中心所引起的大的、不稳定的平涡流层，也可以发现Trump塔楼所引起的小的、不同的涡流。

Conclusions

When performing the aerodynamic analysis of tall buildings, it is important to accurately capture the flow separation due to both the boundary layer and geometric features. The ability to place the buildings in realistic surroundings is also vital for detailed analysis to be possible, both for creating a realistic flow field around an individual building, and also when looking wider afield for master planning projects.

Standard resolution CFD cannot meet the level necessary to adequately capture either form of flow separation, and potentially will lose important geometric features if they are smaller than the level of the mesh. Wind tunnels are necessarily small scale, and are not able to model buildings in the "hyper-critical" Reynolds number regime, which most tall buildings occupy, so may not be able to adequately capture boundary layer induced flow separation. As more and more megatall (600m+) buildings are planned, this situation can be expected to become increasingly exaggerated.

High resolution CFD, which requires an HPC facility such as the one at Wirth Research, is able to accurately capture the effects of both types of flow separation. CFD at this level typically has a resolution of 0.3m cell size on a full-scale tall building, and will place the building in a realistic cityscape of up to 10km diameter.

The level of aerodynamic detail captured by High Resolution CFD will have added benefits in the prediction of pedestrian comfort, as inaccurate prediction of flow separation could suggest that a specific region is located within a wake, rather than in a region of accelerated flow, where the level of comfort is considerably different. The accuracy allowed by High Resolution CFD means that the probability of exceeding specified comfort criteria can be assessed at intervals as fine as 0.1m apart.

If transient High Resolution CFD is used (DES), then there is also the capability of capturing the fluctuating dynamic pressures, which can cause building motions, to the same level of accuracy as the mean building forces have been demonstrated to be predicted in this paper.

结论

当进行超高层建筑的气动学分析，准确的捕捉边界层和几何属性所引起的气流分离是至关重要的。把建筑放在真实城市环境中去分析的能力是十分重要的，它可以做细节的分析，用于模拟建筑周围真实的气流环境，也同时可以广泛的用于城市规划的项目。

标准精度的CFD模拟不能充分满足捕捉两种气流分离的要求，如果建筑细节小于网格大小，模型会失去重要的几何特性。风洞模型往往是小尺寸的，不能模拟很多超高层建筑在跨临界流雷诺数条件下的情况，所以不能充分捕捉边界层的分离。因为越来越多的超高层建筑 (600米以上) 在规划之中，这种需求会越来越大。

高精度CFD需要类似WR公司所拥有的高性能计算机，它可以准确捕捉两类气流分离的影响。这种规模的CFD通常具有0.3米的网格大小去模拟全尺寸的超高层建筑，并且可以把建筑放在最多10公里直径的真实城市环境中。

高精度CFD模拟所能捕捉的气动学细节可以有额外的好处，比如行人的舒适性研究。因为不准确的气流分离可能建议行人处在尾流区，而不只是气流产生区，它们有完全不同的舒适性程度。高精度CFD所能提供的准确性可以在小到0.1米的尺度上预测有没有超过舒适条件。

如果瞬态高精度CFD (DES) 被采用，就会有能力动态的捕捉可能导致建筑晃动的压力变化，精度同样可以达到这篇文章里所描述的平均建筑剪力的程度。

这个工具不但可以用于建筑设计过程中精确的气动学分析，还可以连接建筑本身和大范围的城市规划的策略。这对创造建筑环境的所有参与者都是有好处的。

此文章由英国牛津布鲁克斯大学牛津可持续发展研究中心 / 建筑学院讲师杜琥博士翻译成中文。

This tool can not only be used for precise aerodynamic analysis throughout the design process, but can also be used to link the building into a wide ranging master planning strategy. This can have clear benefits for all parties involved in the creation of the built environment.

English to Chinese translation by Dr H Du, Lecturer in Architecture and Energy Simulation, Low Carbon Building Group, School of Architecture / Oxford Institute for Sustainable Development, Oxford Brookes University

References (参考书目):

Baker, W., James, P., Tomlinson, R. and Weiss, A. (2009), "**Case Study: Trump International Hotel and Tower**", CTBUH Journal 2009, Issue III, pp16-22.

Council on Tall Buildings and Urban Habitat (CTBUH), (2014), "**2013: A Tall Building Review**", CTBUH Journal 2014, Issue I, pp1-2.

Franke, J., Hellsten, A., Schlünzen, H. and Carissimo, B. (2007), "**Best Practice Guideline for the CFD Simulation of Flows in the Urban Environment**", COST Action 732.

Jones, G., Cincotta, J. and Walker, R. (1969), "**Aerodynamic Forces on a Stationary and Oscillating Circular Cylinder at High Reynolds Numbers**", NASA Technical Report TR R-300.

Obasaju, E. (1992), "**Measurement of Forces and Base Overturning Moments on the CAARC Tall Building Model in a Simulated Atmospheric Boundary Layer**", Journal of Wind Engineering and Industrial Aerodynamic, vol. 4, pp103-126.

Polhamus, E. (1984), "**A Review of Some Reynolds Number Effects Related to Bodies at High Angles of Attack**", NASA Contractor Report 3809.