Design Problem 10

Design of Industrial Fan Blades

(Design Problem Courtesy of Halifax Fan Ltd)

KEY FEATURES INTRODUCED IN THIS DESIGN PROBLEM

	Key features
1	Cyclic Symmetry
2	Manual Convergence of Results

INTRODUCTION

Halifax Fan Ltd is one of the world's foremost manufacturers of industrial fans who design and manufacture a full range of centrifugal fans from a wide range of materials including mild and stainless steel, from their manufacturing operations in the UK and China. Industrial customers supplied are wide ranging and include power, pharmaceuticals, chemical, nuclear, and marine markets all over the world.

doi:10.1016/B978-1-85617-694-1.00012-3

Halifax Fan is fully BSI certified to BS EN ISO9001 – 2000 and manufactures fans to many industrial standards including API 673, API 560, Shell DEP, and ATEX among others. Many of these designs are engineered to meet the customer's exact requirements and thus offer a wide range of services on- and off-site including stress relieving, laser shaft alignment, site performance testing, vibration analysis, consultation, problem solving, repairs, and energy testing. As a consequence of offering special bespoke solutions, Halifax is regularly asked by its customers to validate their designs prior to delivery.

Some of the typical requirements include determining the following:

1. Maximum stress and deflection of the fan blade.
2. Factor of safety of the new design.

In addition to the above requirements, the design criteria to be used for this design problem are as follows:

- Material to be used is either mild steel or steel – high strength low alloy[1]
- Factor of safety required is 1.75
- Maximum deflection is 0.5 mm
- Maximum blade thickness is 5 mm

WORKFLOW OF DESIGN PROBLEM 10

---

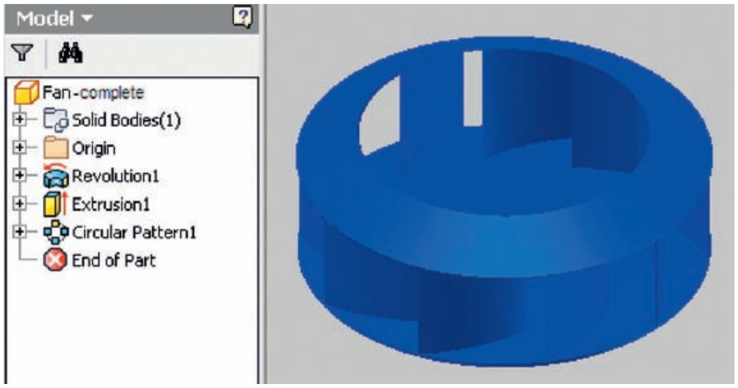
[1] Halifax Fan actually uses Carbon Steel to BS EN 10025 grade S275JR for its fans.

Idealization

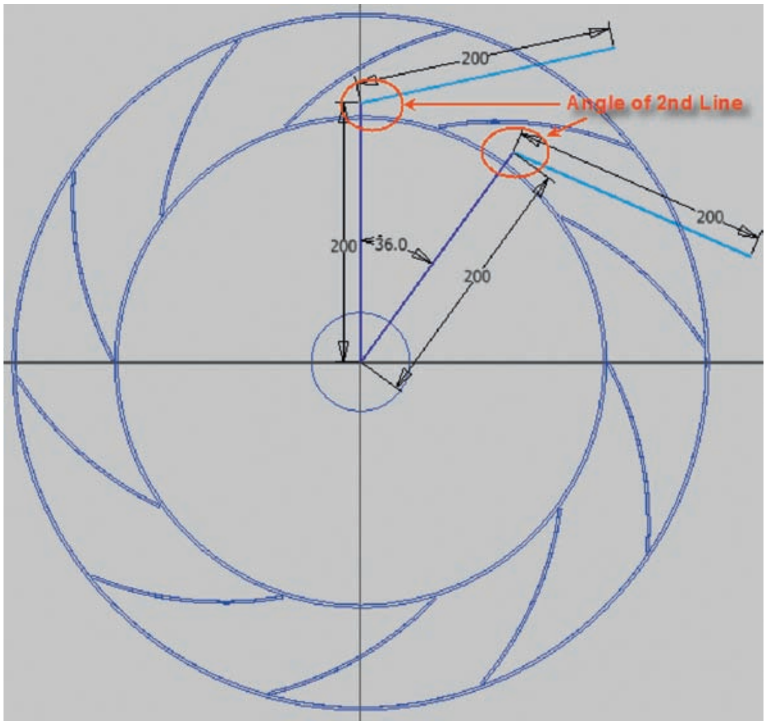
Halifax Fans can range from simple small fans to large detailed fans. In the cases of large detailed fans, the size of the mesh can become very large and the time taken to analyze results can become very lengthy.

Most fans comprise a number of similar blades and when in operation, the deflection and stress induced in the blades are identical and for this reason it is only necessary to analyze one blade of the fan. This simplification approach is also referred to as cyclic symmetry and significantly reduces the model size giving more scope to refine and analyze the results efficiently. Therefore, in the following steps, the Fan model is split such that only one blade remains.

1. Open *Fan-complete*.ipt



2. Create new Sketch on YZ plane to the following dimensions



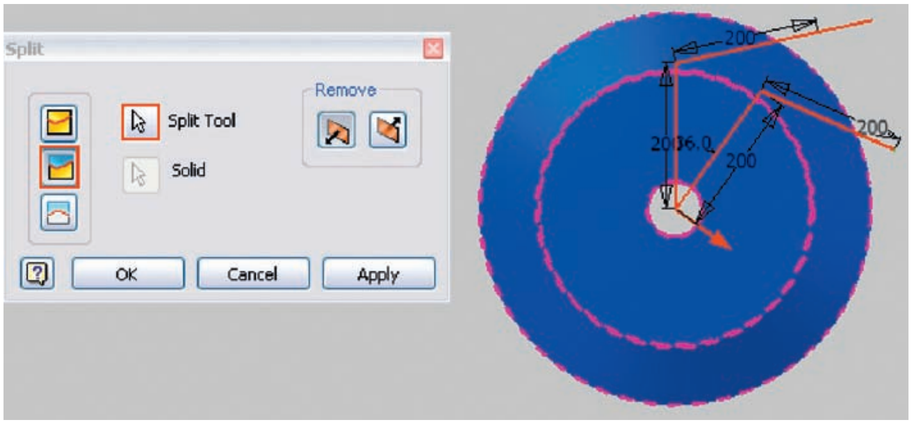
It will help to change the model display to wireframe or transparent color when creating the Sketch as it will allow you to see the top plate.

As there are 10 blades, we need to split the model by 36 degrees angle.

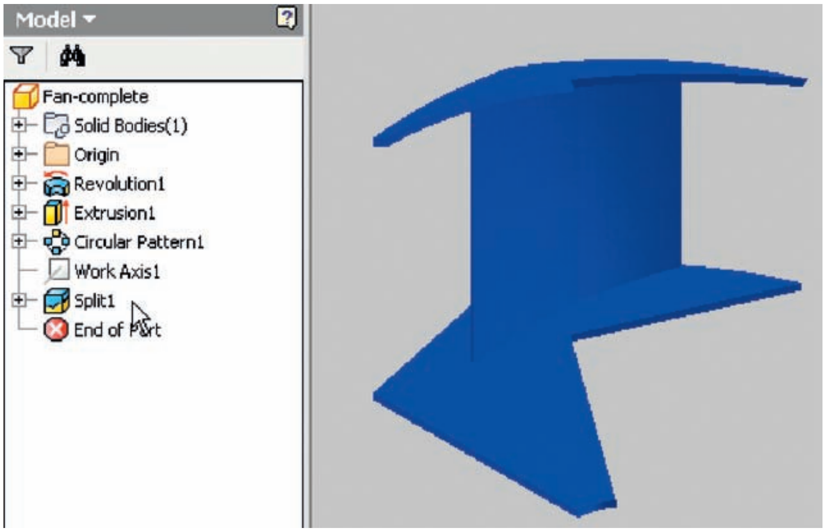
$$\text{Angle of Split to Create Single Blade} = \frac{360}{\text{Number of Blades}} = \frac{360}{10} = 36 \text{ degrees}$$

The angle of the second line is not critical as long as the line is more or less positioned in the middle of two blades.

3. Finish Sketch > Using the split feature, split the part using the sketch created



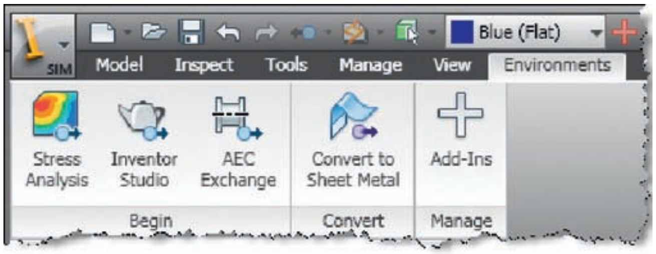
4. Click OK



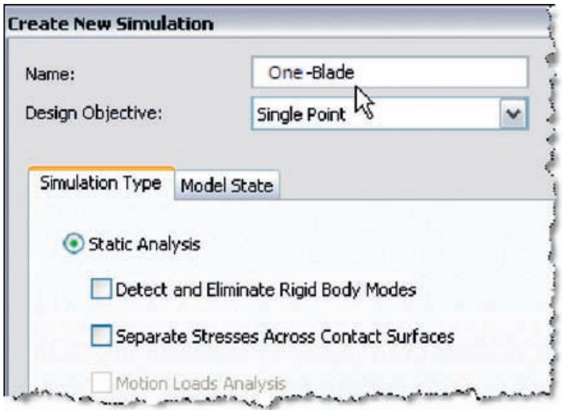
Now, in the next section, the boundary conditions will be applied to the single fan blade.

Boundary conditions

5. Select Environments tab > Stress Analysis



6. Select Create Simulation > Specify One-Blade for Name > Click OK


Create New Simulation

Name: One -Blade

Design Objective: Single Point

Simulation Type | Model State

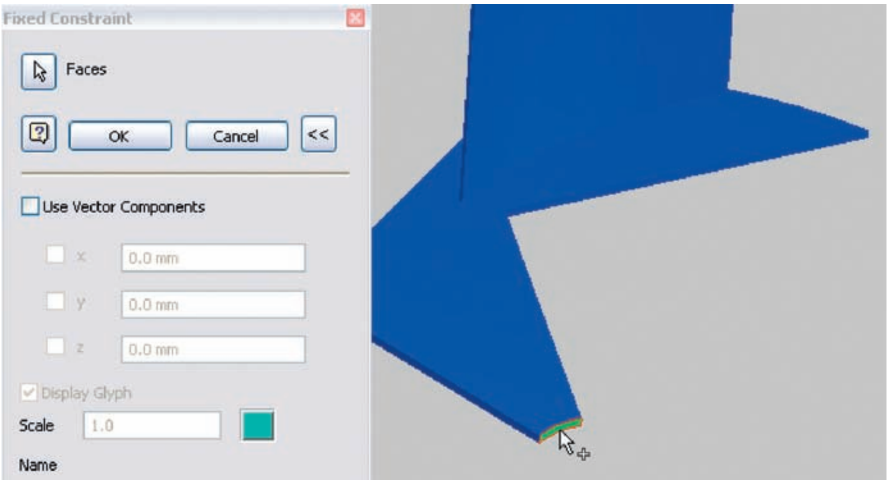
● Static Analysis

☐ Detect and Eliminate Rigid Body Modes

☐ Separate Stresses Across Contact Surfaces

☐ Motion Loads Analysis

7. Select Fixed Constraint > Select face as shown > Click OK


Fixed Constraint

Faces

☐ Use Vector Components

☐ x 0.0 mm

☐ y 0.0 mm

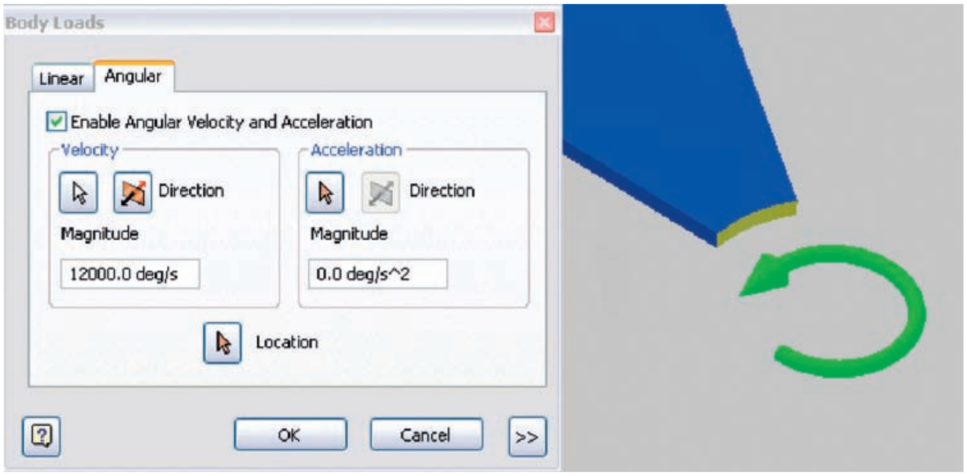
☐ z 0.0 mm

☑ Display Glyph

Scale 1.0

Name

8. Select Body Loads > Select Angular Tab > Enable Angular Velocity > Select face to specify direction of fan speed > Specify 2000 rpm > Click OK


Body Loads

Linear | Angular

☑ Enable Angular Velocity and Acceleration

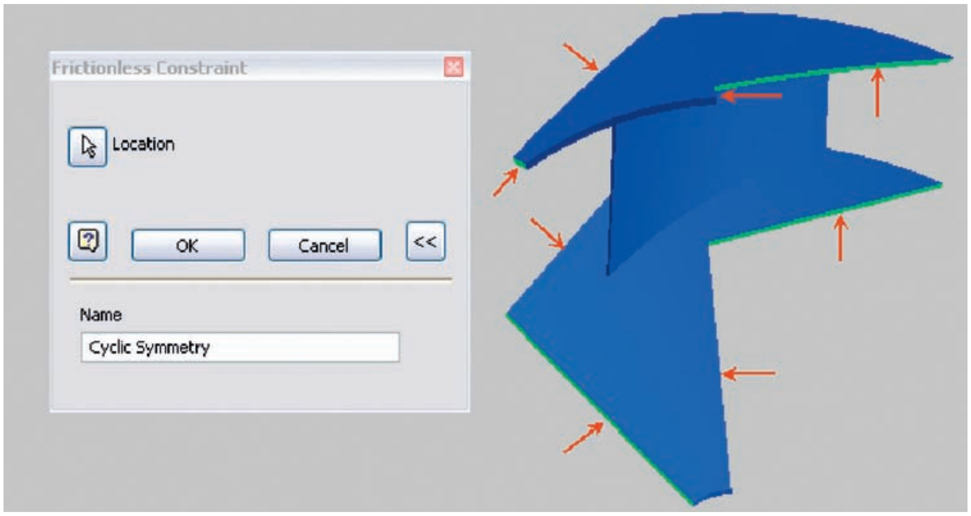
Velocity: Direction; Magnitude: 12000.0 deg/s

Acceleration: Direction; Magnitude: 0.0 deg/s^2

Specifying revolutions per minute after the value will convert the value to the default degrees/seconds.

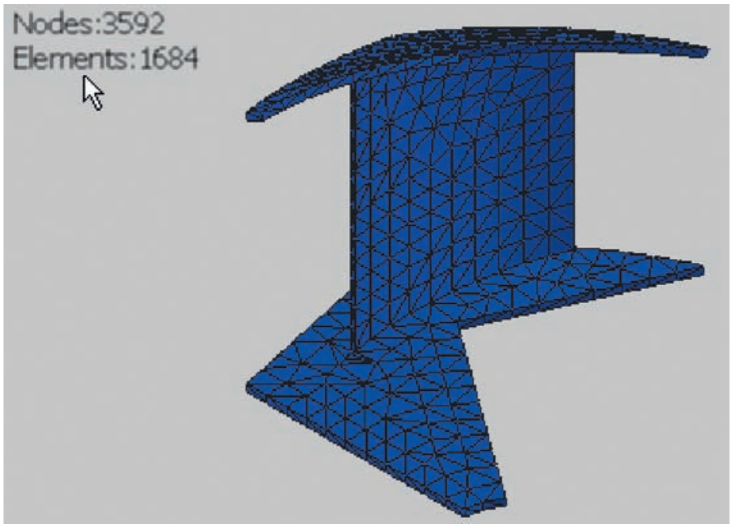
If the complete fan was analyzed, the boundary conditions specified in steps 7 and 8 would suffice. However, as we are only modeling a single blade, we need to specify extra boundary conditions to enable it to behave like a complete model. This can be achieved by applying frictional constraints on all faces that are created as a result of the split feature.

9. Select Frictionless Constraint > Select all 8 faces on the split planes > Specify Cyclic Symmetry for Name > Click OK

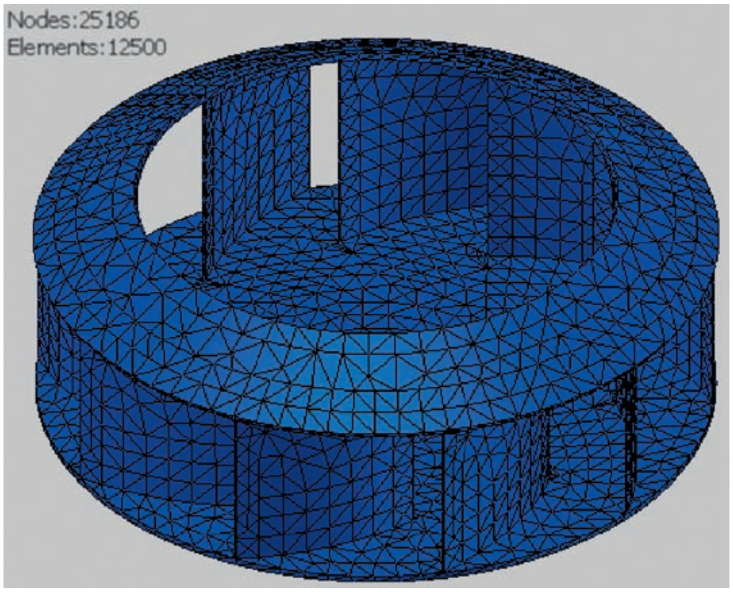

Frictionless Constraint

Name: Cyclic Symmetry

10. Select Mesh View


Nodes:3592
Elements:1684

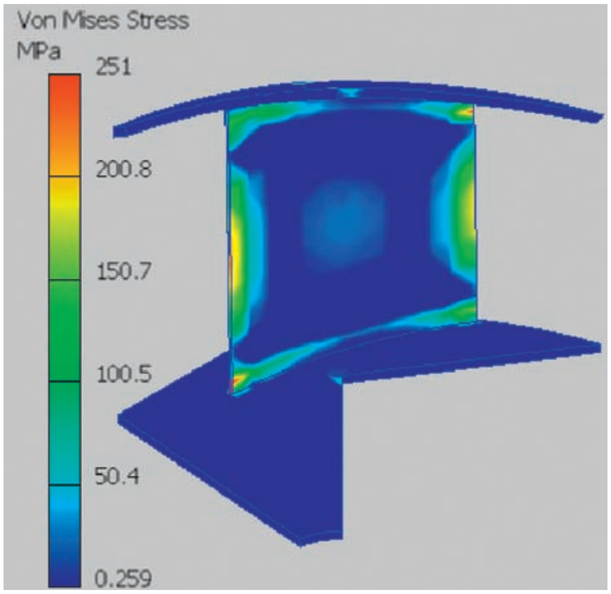
A complete model would create many more elements as illustrated below.


Nodes:25186
Elements:12500

Run simulation and analyze

11. Select Simulate > Run Analysis

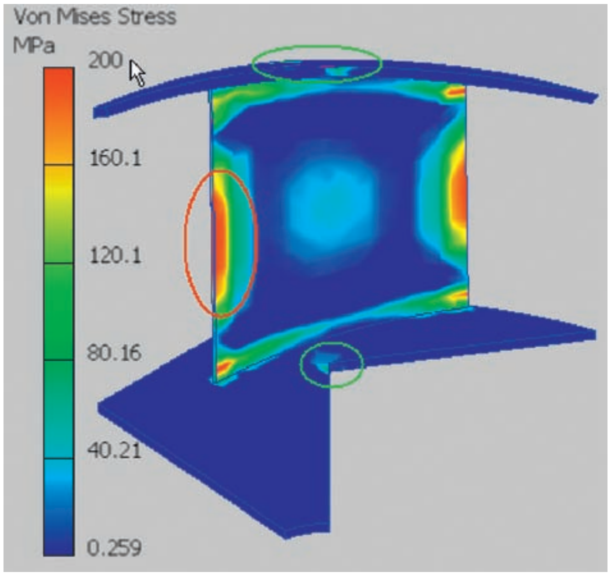
12. Select Actual for Displacement Scale > Deselect Mesh View



Stress singularities will appear at the blade and plate interface due to sudden geometrical discontinuities and will be ignored as the area of interest is in the middle of the blades.

Stress singularites may also occur in the area of the split faces and can be ignored as they would have not appeared if the complete fan were analyzed.

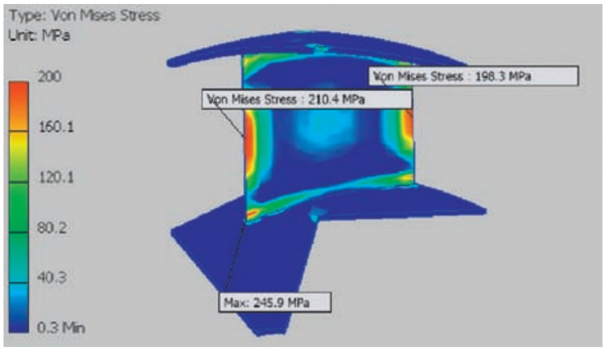
13. Select Color Bar > Unselect Maximum > Specify 200 MPa > Click OK



Use the color bar to pinpoint stress display in the area of interest and to enhance stress display.

As we are interested in the middle of the blade, we can use probe to display stresses in the area of interest to us.

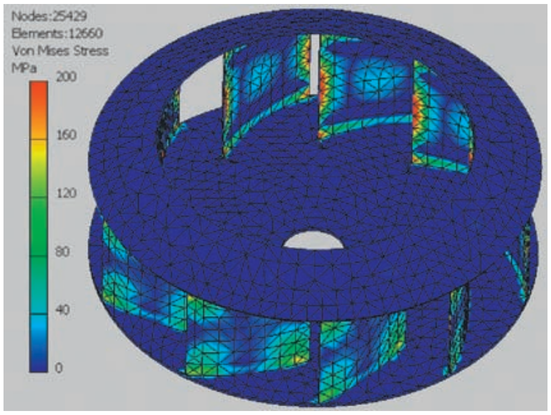
14. Select Probe > Select in the middle of the blade at the front and rear



Zoom into the area of interest before selecting the area of interest using probe.

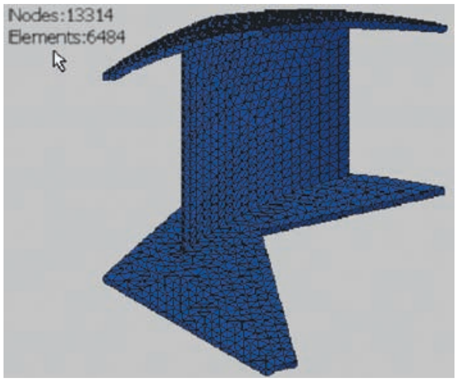
IMPORTANT—Extract stress value of probe is dependant on location clicked, hence value may slightly differ.

Below is a stress plot of a complete model, illustrating similar stresses in all the blades of the fan.



Now, we will increase the mesh to see if the stress results change in the blades.

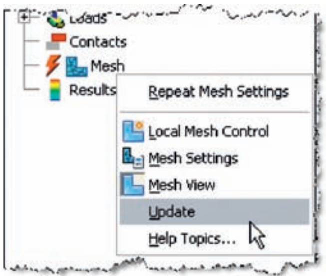
15. Select Mesh Settings > Change Average Element Size to 0.05 > Click OK



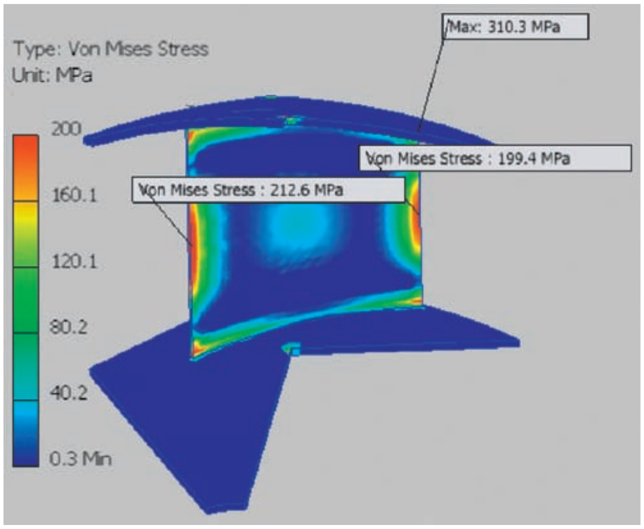
Reducing the average element size can have a significant impact on the size of the mesh.

Reducing the average element size from 0.1 to 0.05 has increased the number of elements by 285% and will thus take longer to run the simulation.

16. Right Click Mesh > Select Update Mesh



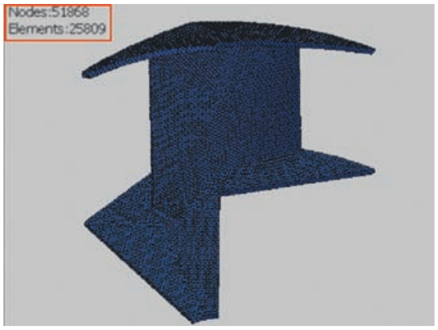
17. Rerun Simulation



Although the maximum stress has moved to the back of the blade and increased by 26.5%, the stress in the middle of the blade has only changed by 1% to 212.6 MPa. Use your probe value for comparison.

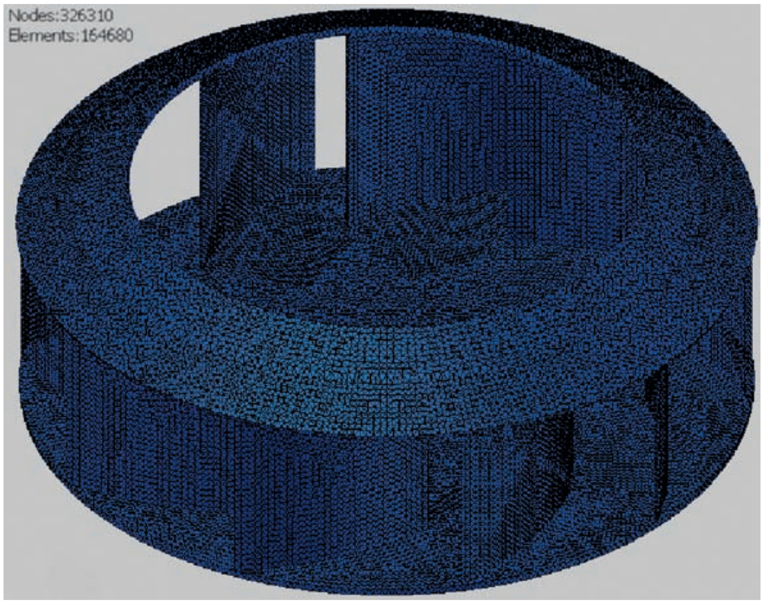
To confirm whether this stress in the middle of the blade has converged, we will rerun one more analysis with a smaller element size.

18. Select Mesh Settings > Change Average Element Size to 0.025 > Click OK

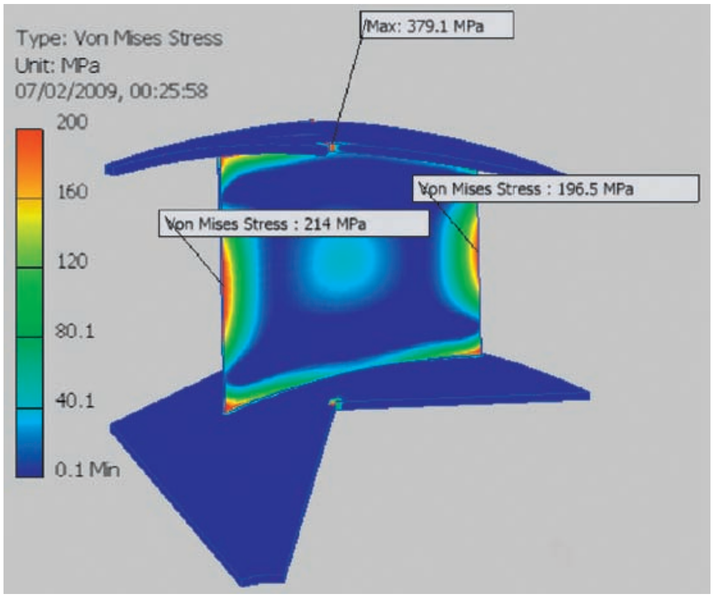


Reducing the average element size from 0.1 to 0.025 has increased the number of elements by 1,432%.

A full model with similar mesh size of 0.025 will create 164,680 elements as illustrated below.

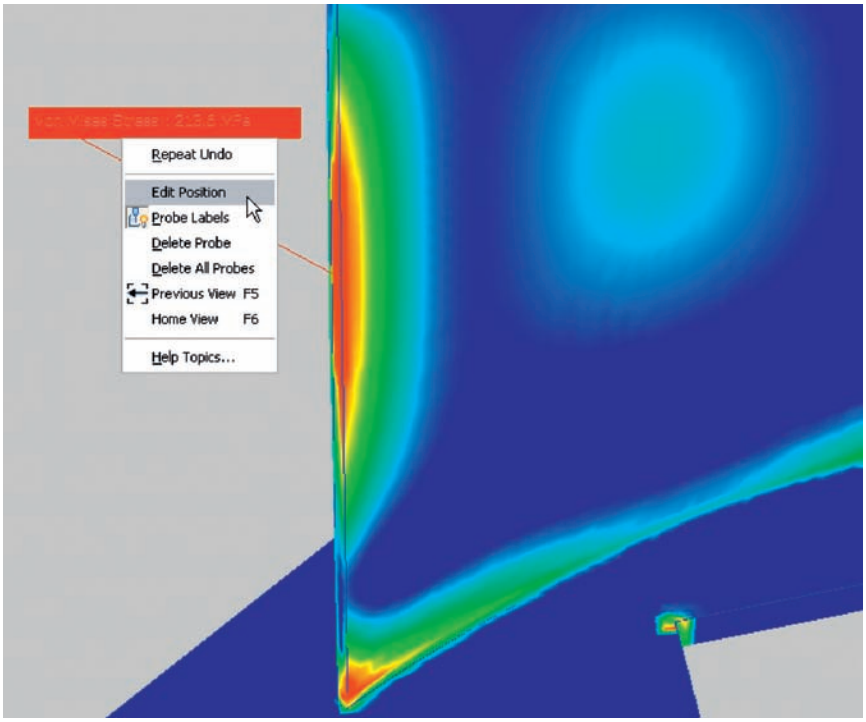


19. Right Click Mesh > Select Update Mesh > Rerun simulation



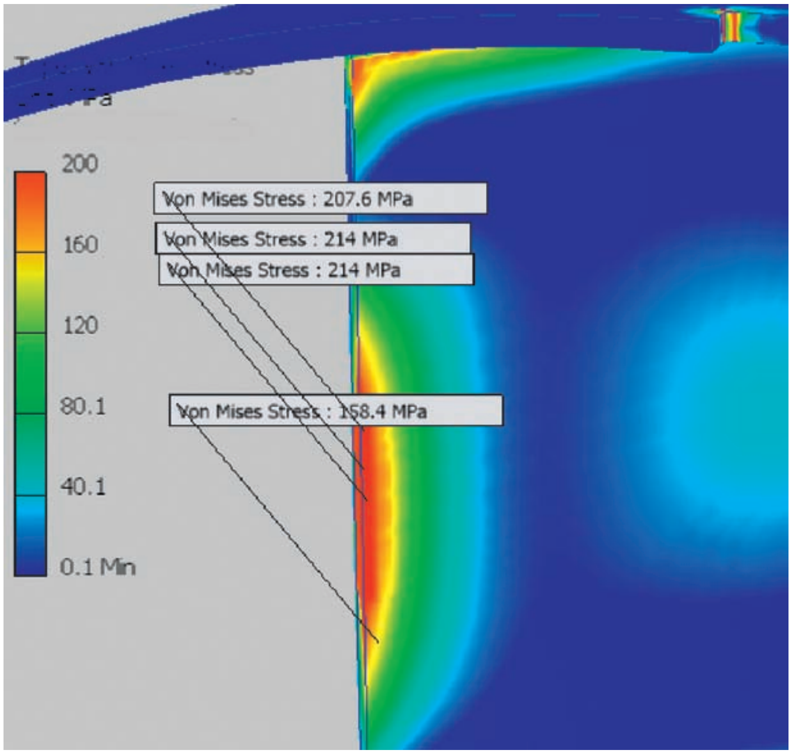
Ignore maximum stress as it is occurring on the top plate and blade interface due to geometrical discontinuities leading to stress singularities.

20. Right Click Probe > Select Edit Position



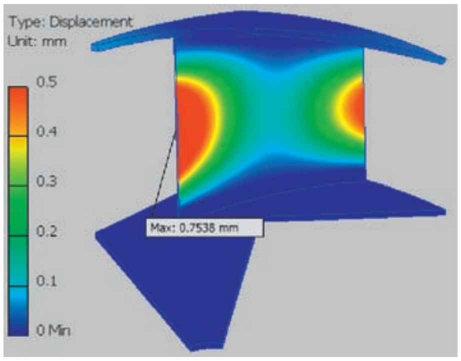
By changing the position, you can display results in different areas of the model.

Alternatively, you can select multiple areas of the model with the probe option.

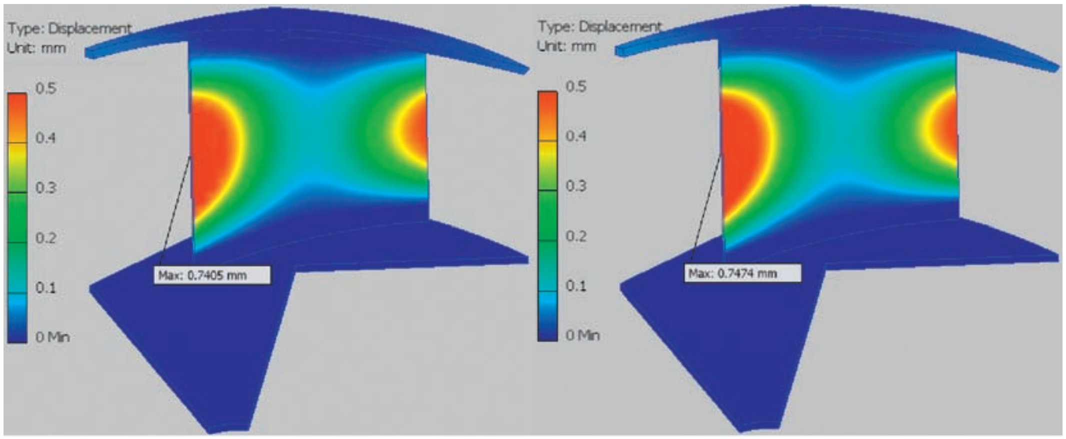


The maximum value in the middle of the blade does not exceed 218 MPa. As we are only interested in this region, we could confidently say that the results have converged in the area of interest.

21. Double Click Displacement from the Stress Analysis browser

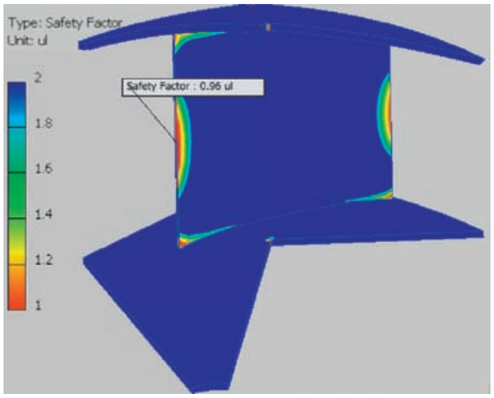


The maximum displacement plots for mesh settings of 0.1 and 0.05 are also shown below.



The maximum displacement occurs in the middle of the blade and changes from 0.7405 to 0.7538, a change of 1.8% such as the displacement values can also be treated as having converged. Values may differ slightly.

22. Double Click Safety Factor



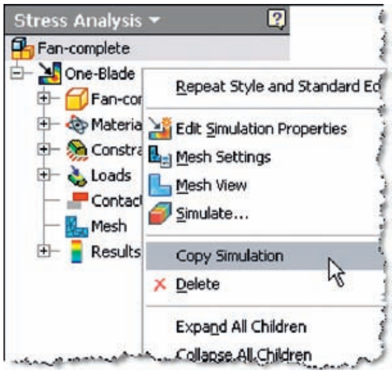
Use the color bar to adjust range.

Based on the stress in the middle of the blade (214 MPa), we have a safety factor below 1, which suggests that the design has failed as the design limit was 1.5. In the next section, we will perform an optimization study to meet the design limits.

Optimization

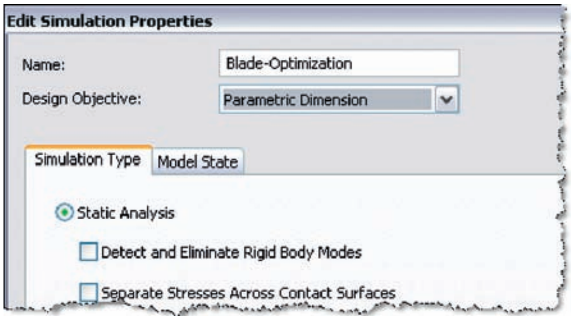
In this section, we will alter blade thickness from 2 mm to 5 mm using the parametric study and manually alter the material from Mild Steel to High Strength Steel.

23. Right Click One-Blade > Select Copy Simulation > Click OK



24. Right Click copied Simulation:1 > Select Edit Simulation Properties

25. Specify Blade-Optimization for Name > Select Parametric Dimension for Design Objective > Click OK

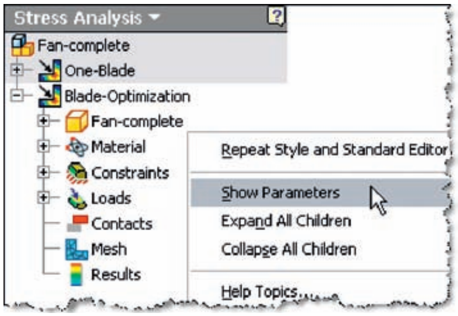

Edit Simulation Properties

Name: Blade-Optimization

Design Objective: Parametric Dimension

This will now allow us to carry out a parametric study.

26. Right Click Fan-complete.ipt in the browser > Show Parameters

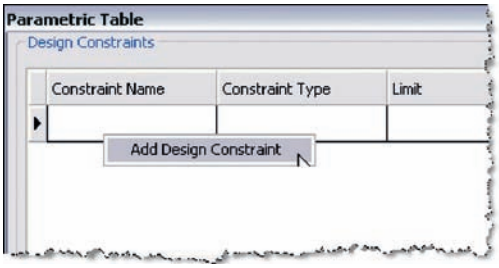


27. Select Bladethickness User Parameter > Click OK

Reference Parameters				
	d13	mm	266.5 mm	266.5
User Parameters				
☑	Bladethickness	mm	2 mm	2.0

28. Select Parametric Table

29. Right Click in Design Constraints row > Select Add Design Constraint

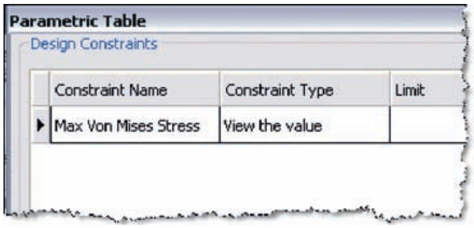


Parametric Table

Design Constraints

Constraint Name	Constraint Type	Limit

30. Select Von Mises from the list



Parametric Table

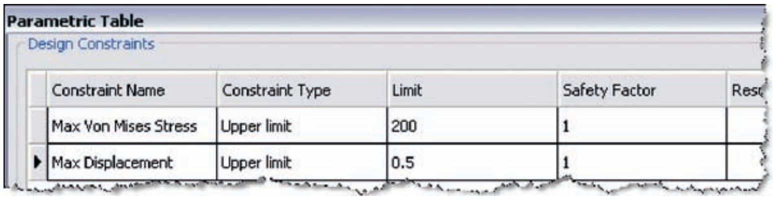
Design Constraints

Constraint Name	Constraint Type	Limit
Max Von Mises Stress	View the value	

31. Repeat step 29 to add Displacement Design Constraints

32. Change the Constraint Type for Max Von Mises Stress to Upper limit > Specify Limit to be 200

33. Change the Constraint Type for Max Displacement to Upper limit > Specify Limit to be 0.5

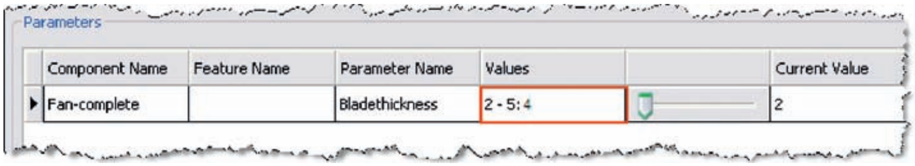


Parametric Table

Design Constraints

Constraint Name	Constraint Type	Limit	Safety Factor	Res
Max Von Mises Stress	Upper limit	200	1	
Max Displacement	Upper limit	0.5	1	

34. Specify 2 - 5:4 in the Bladethickness Value field



Parameters

	Component Name	Feature Name	Parameter Name	Values		Current Value
▶	Fan-complete		Bladethickness	2 - 5: 4		2

This will generate values of 2, 3, 4, 5 such as will create three additional parameters.

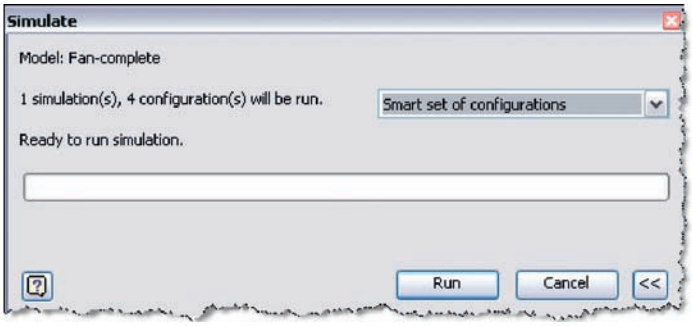
35. Right Click anywhere in the Parameter Rows and select Generate Range Configurations

36. Move the slider to see the blade changing its thickness > Click Close

37. Select Mesh Settings > Specify 0.05 Average Element Size

38. Select Mesh View

39. Select Simulation > Run Simulation



Simulate

Model: Fan-complete

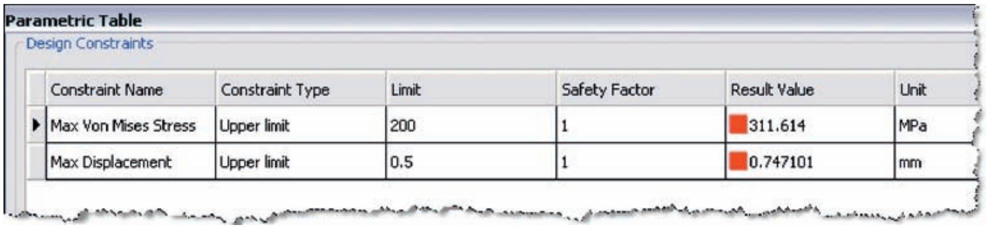
1 simulation(s), 4 configuration(s) will be run. Smart set of configurations

Ready to run simulation.

40. Select Actual for Displacement Scale

41. Select Parametric Table

The red icon indicates unacceptable parameters based on Constraint Limits.



Parametric Table

Design Constraints

	Constraint Name	Constraint Type	Limit	Safety Factor	Result Value	Unit
▶	Max Von Mises Stress	Upper limit	200	1	🟥311.614	MPa
	Max Displacement	Upper limit	0.5	1	🟥0.747101	mm

The Max Von Mises Stress value is also misleading as this value represents stress singularities in the model. To synchronize the Stress Limit with the Model, change the color bar range between 0 and 200.

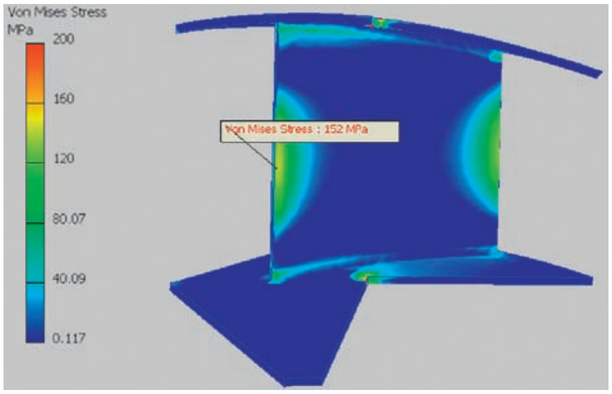
42. Select Color bar > Specify 200 for Maximum value > Click OK

Now move the slider and compare the color plots as you move the slider between 2 and 5. From the color plots, blade thickness values 4 and 5 do not show any red color in the blades indicating low stress, with thickness 5 showing least stresses.

43. Move the slider to read a value of 5 > Select Close

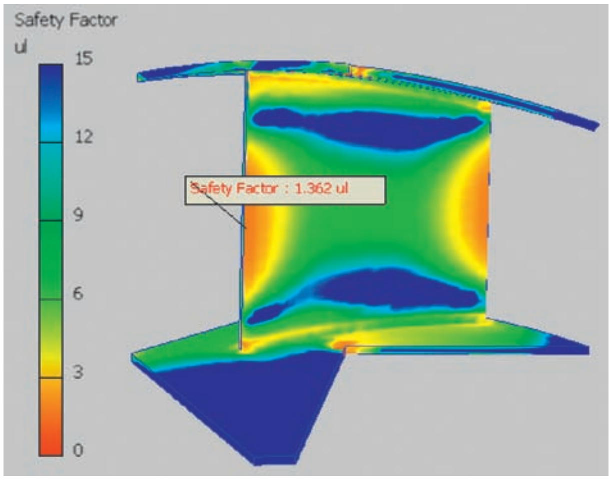
We will now use the probe to determine the exact value of stress in the middle of the blade.

44. Select Probe and select Blade at the highest stress point



You may need to select several locations to get an indication of the highest stress point.

45. Double Click Safety Factor



The safety factor is still below the design limit of 1.5, so we will now assign a new material.

46. Select Assign Material

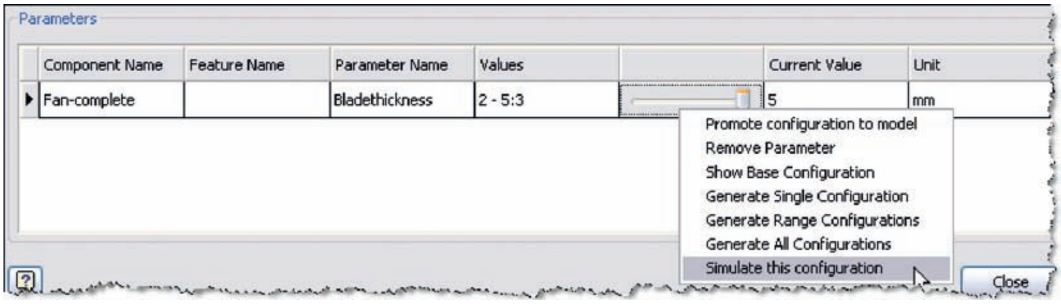
47. Select Steel, High Strength Low Alloy from the Override Material list > Click OK

Assign Materials

Component	Original Material	Override Material	Safety Factor
Fan-complete	Steel, Mild	Steel, High Strength Low	Yield Strength

48. Select Parametric Table > Move Slider to read current value of 5

49. Right Click slider > Select Simulate this configuration > Select Run

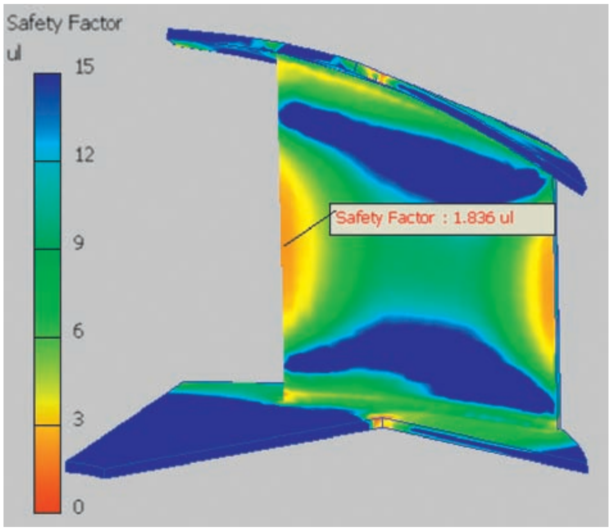


Parameters

	Component Name	Feature Name	Parameter Name	Values		Current Value	Unit
▸	Fan-complete		Bladethickness	2 - 5:3		5	mm

50. Select Actual for Displacement Scale > Double Click Safety Factor

51. Select Probe > Select area of minimum safety factor



Now by changing the material, we have reached our goal of having a safety factor above 1.5 and a maximum displacement below 0.5 mm. Ignore max stress as it is occurring on the top plate, in reality this does not exist. Refer to the stress display of the complete fan earlier.

52. Close the file