Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 112 (2017) 344 – 351

Energy Procedia

Sustainable Solutions for Energy and Environment, EENVIRO 2016, 26-28 October 2016,
Bucharest, Romania

Numerical Study of a Gas Turbine's Shaft Cooling

Laura Alina STIKA[a]*, Gheorghe MEGHERELU[a], Valeriu Alexandru VILAG[a]

[a]*Romanian Research and Development Institute for Gas Turbines COMOTI, Bucharest, Romania*

Abstract

This paper focuses on the numerical study of a shaft's cooling, positioned at the end of the gas turbine hot section. Due to convection and conduction phenomena, the high temperatures of the primary gas flow are transferred -via the power turbine's blades and disk- to the power turbine's shaft. If the values of these transmitted temperatures are too high, in time, negative effects can occur, like: rotor imbalance or overheating the oil surrounding the power turbine's shaft. Therefore, for the turbine's shaft to operate properly, a cooling method must be applied. The cooling method considered in this study, is a secondary air flow path, which crosses through the labyrinth, part of the shaft itself and turbine's disk.

In order to investigate the heat transfer between the secondary air flow and the turbine's shaft, several computations were performed, using the CFD software Ansys CFX. Several cases were analyzed, by modifying the geometry, the material, as well as the air mass flow passing through the analyzed path. The comparison between the different results led to the selection of the best configuration, in terms of cooling performances.

Peer-review under responsibility of the organizing committee of the international conference on Sustainable Solutions for Energy and Environment 2016

Keywords: Heat Transfer; CFD; Cooling, Axial Turbine.

---

1. Introduction

It is known that in order to have good performances, a gas turbine needs to operate at increased temperatures.

---

* Corresponding author. Tel.: +4-021-434-0236; fax: +4-021-434 02 41.
E-mail address: laura.stika@comoti.ro

Peer-review under responsibility of the organizing committee of the international conference on Sustainable Solutions for Energy and Environment 2016
doi:10.1016/j.egypro.2017.03.1063

Although, increasing the firing temperature in a turbo shaft has a positive and very predominant effect in terms of work output and thermal efficiency[1] it could have a negative impact on its various components. A great thermal gradient on a power turbine's shaft could lead to some unwanted effects, including rotor imbalance (due to uneven dilatation of unevenly heated areas) [2]. On the other hand, the shaft's temperature is also transferred to the oil that surrounds it. If this temperature is greater than the oil's admissible one, it will have negative consequences on the lubrication proprieties and could damage the integrity of the secondary gas turbine components such as seals, disks, bearings and shafts. Hence it is important to control the peak temperature and temperature distribution on these components. This is can be accomplished by the application of additional cooling solutions to the high temperature structures [3]. This paper focuses on one of these solutions, which consists in developing a secondary air flow path, which crosses through the labyrinth, part of the shaft itself, an internal profile cap and turbine's disk. The schematics and position of the application is presented in Fig. 1

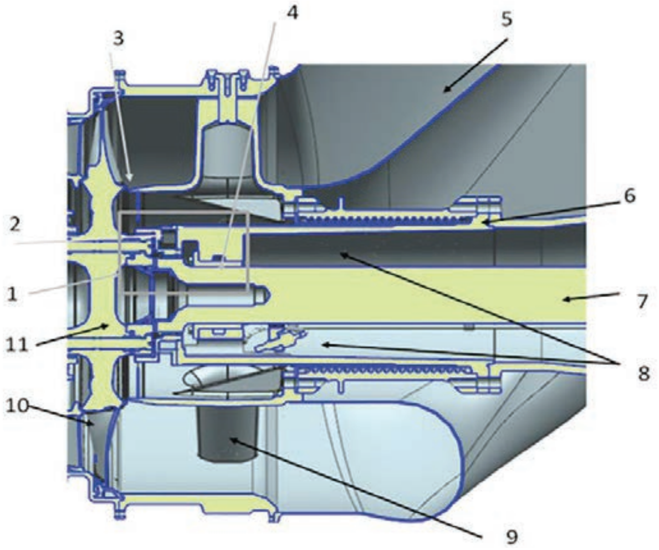

Fig. 1. Schematics of the secondary cooling air path and its position.
1-Location of the secondary air path; 2- Labyrinth and the Inlet in the secondary air path; 3- Exit of the secondary air path; 4-"Measure wall's" position; 5- Exhaust system of the turbo shaft; 6- The shaft's casing; 7- The power turbine's shaft; 8- Region with oil, surrounding the shaf; 9- Strut; 10- Power turbine's blade; 11- Power turbine's disk.

The goal of the work presented herein is to study the heat transfer between this secondary air path and the power turbine's shaft.

2. Method

To achieve this goal, a set of 9 simulations of the flow inside the secondary air path presented in Fig. 1 was carried out, using the CFD software Ansys CFX. The configurations used for the computations are built on:

- Three types of geometry
- Three inlet pressures
- Two types of steel
- Non-rotating or rotating fluid and solid computational domain.

The numerical simulations provided the temperature on the shaft, the temperature distribution on the domain walls and also information about the flow inside this secondary path. The maximum temperature and its distribution on the "measure wall" - in Fig. 1 - were considered to best describe the cooling performance of secondary air path.